OOFEM Element Library Manual

Bořek Patzák

Czech Technical University
Faculty of Civil Engineering
Department of Structural Mechanics
Thákurova 7, 166 29 Prague, Czech Republic

May 21, 2026

Contents

List of Figures

1 Introduction

In this manual the detailed description of available elements is given. The actual availability of particular elements depends on OOFEM configuration. Elements are specified using element records, which are part of oofem input file. The general format of element record is described in OOFEM input manual.

Every element is described in a separate section. The section includes the "element keyword", which determines the element type in element record, approximation and interpolation characteristics, required cross section properties (which are summarized in "CS properties" part), and a summary of element features. The "Load" section contains useful information about the types of loadings supported by particular elements.

2 Elements for Structural Analysis (SM Module)

2.1 Truss Elements

2.1.1 Truss 1D element

Represents linear isoparametric truss element in 1D. The elements are assumed to be located along the x-axis. Requires cross section area to be specified. The element features are summarized in Table 1.

Keyword	**truss1d**
Description	1D truss element
Specific parameters	-
Unknowns	Single dof (u-displacement) is required in each node
Approximation	Linear approximation of displacement and geometry
Integration	Exact
Features	Full dynamic analysis support, Full nonlocal constitutive support, Adaptivity support
CS properties	Area is required
Loads	Body loads are supported. Boundary loads are not supported in current implementation
Status	Reliable

Table 1: truss1d element summary

2.1.2 Truss 2D element

Two node linear isoparametric truss element for 2D analysis. The element geometry can be specified in (x,z), (x,y), or (y,z) plane. The element features are summarized in Table 2.

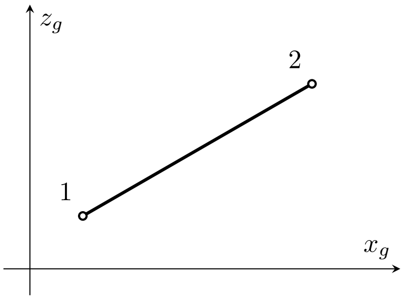

Figure 1: Truss2d element in (x,z) plane.

truss2d element summarytruss2dsummary

2.1.3 Truss 3D element

Two node linear isoparametric truss element for 3D analysis. The element geometry is specified in (x,y,z) space. The element features are summarized in Table 3.

Keyword	**truss2d**
Description	2D truss element
Specific parameters	[`cs` #$_{\text{(in)}}$]
Parameters	`cs`: this parameter can be used to change default definition plane. The supported values of `cs` are following: 0 for (x,z) plane (default), 1 for (x,y) plane, and 3 for (y,z) plane.
Unknowns	Two dofs representing displacements in definition plane are required in each node. The element can be used in different planes, default definition plane is (x,z). The parameter `cs` can be used to change default definition plane.
Approximation	Linear approximation of displacements and geometry.
Integration	Exact.
Features	Full dynamic analysis support. Full nonlocal constitutive support.
CS properties	cross section area should be provided.
Loads	Edge loads are supported, Edge number should be equal to 1
Status	Reliable

Table 2: truss2d element summary

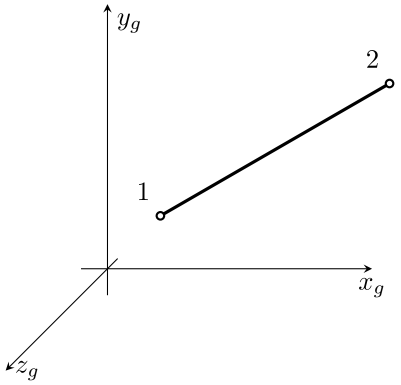

Figure 2: Truss3d element in (x,y,z) space.

Keyword	**truss3d**
Description	3D truss element
Specific parameters	-
Unknowns	Three displacement DOFs (in x, y, and z directions) are required in each node.
Approximation	Linear approximation of displacements and geometry.
Integration	Exact.
Features	Full dynamic analysis support. Full nonlocal constitutive support.
CS properties	cross section area should be provided.
Status	Reliable

Table 3: truss3d element summary

2.2 Beam Elements

2.2.1 Beam2d element

Beam element for 2D analysis, based on Timoshenko hypothesis. Structure should be defined in x,z plane. The internal condensation of arbitrary DOF is supported and is performed in local coordinate system. On output, the local end displacement and local end forces are printed. The element features are summarized in Table 4.

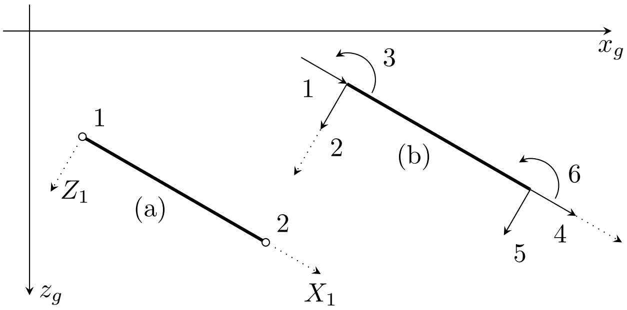

Figure 3: Beam2d element. Definition of local c.s.(a) and definition of local end forces and local element dofs (b).

Keyword	**beam2d**
Description	2D beam element
Specific parameters	[`dofstocondense` #$_{\text{(ia)}}$]
Parameters	`dofstocondense`: allows to specify local element dofs that will be condensed. The numbering of local element dofs is shown in fig. 3. The size of this array should be equal to number of local element dofs (6) and nonzero value indicates the corresponding dof will be condensed.
Unknowns	Three dofs (u-displacement, w-displacement, y-rotation) are required in each node.
Approximation	Cubic approximations of lateral displacement and rotation are used. For longitudinal displacement the linear one is assumed.
Integration	Exact.
Features	Full dynamic analysis support. Linear stability analysis support.
CS properties	Area,inertia moment along y-axis (`iy` parameter) and equivalent shear area (`sheararea z` parameter) should be specified.
Loads	Constant and linear edge loads are supported, shear influence is taken into account. Edge number should be equal to 1. Temperature load is supported, the first coefficient of temperature load represent mid-plane temperature change, the second one represent difference between temperature change of local z+ and local z- surfaces of beam (in local coordinate system). Temperature load require that the “thick” property of cross section model is defined.
Status	Reliable

Table 4: beam2d element summary

2.2.2 Beam3d element

Beam element for 3D **linear** analysis, based on Timoshenko hypothesis. The internal condensation of arbitrary DOF is supported and is performed in local coordinate system. On output, the local end-displacement and local end-forces are printed. Requires the local coordinate system to be chosen according to main central axes of inertia. Local element coordinate system is determined by the following rules:

1. let first element node has following coordinates (x_i, y_i, z_i) and the second one (x_j, y_j, z_j),
2. direction vector of local x-axis is then $\mathbf{a}_1 = (x_j - x_i, y_j - y_i, z_j - z_i)$,
3. local y-axis direction vector lies in plane defined by local x-axis direction vector $(\mathbf{a}_1)$ and given point (k-node with coordinates (x_k, y_k, z_k)) - so called reference node,
4. local z-axis is then determined as vector product of local x-axis direction vector $(\mathbf{a}_1)$ by vector $(x_k - x_i, y_k - y_i, z_k - z_i)$,
5. local y-axis is then determined as vector product of local z-axis direction vector by local x-axis direction vector.

The element features are summarized in Table 5.

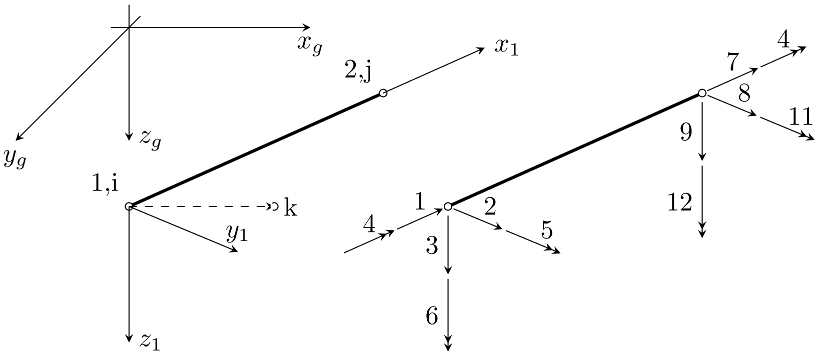

Figure 4: Beam3d element. Definition of local c.s., local end forces and local element dofs numbering.

Keyword	**beam3d**
Description	3D beam element
Specific parameters	`refnode` $\#_{\mathrm{(in)}}$ [`dofstocondense` $\#_{\mathrm{(ia)}}$]
Parameters	`refnode`: sets reference node to determine the local coordinate system of element. `dofstocondense`: allows to specify local element dofs that will be condensed. The numbering of local element dofs is shown in fig. 4. The size of this array should be equal to number of local element dofs (12) and nonzero value indicates the corresponding dof will be condensed.
Unknowns	Six dofs (u,v,w-displacements and x,y,z-rotations) are required in each node.
Approximation	Cubic approximations of lateral displacement and rotation (along local y,z axes) are used. For longitudinal displacement and the rotation along local x-axis (torsion) the linear approximations are assumed.
Integration	Exact.
Features	Full dynamic analysis support. Linear stability analysis support.
CS properties	Area, inertia moment along y and z axis (`iy` and `iz` parameters), torsion inertia moment (`ik` parameter) and either cross section area shear correction factor (`beamshearcoeff` parameter) or equivalent shear areas (`sheareareay` and `shearareaz` parameters) are required. These cross section properties are assumed to be defined in local coordinate system of element.
Loads	Constant and linear edge loads are supported. Edge number should be equal to 1. Temperature load is supported, the first coefficient of temperature load represent mid-plane temperature change, the second one represent difference between temperature change of local z+ surface and local z- surface surface of beam and the third one represent difference between temperature change of local y+ surface and local y- surface of beam. Requires the “thick” (measured in direction of local z axis) and “width” (measured in direction of local y axis) cross section model properties to be defined.
Status	Reliable

Table 5: beam3d element summary

2.3 Lattice elements

2.3.1 Lattice2d element

Represents two-node lattice element for small rotations. Each node has 3 degrees of freedom. The element is based on the Rigid Body Spring Model originally developed by Kawai and later delveloped by Bolander for modelling fracture in concrete. The main idea is to model the elastic and inelastic response of a connection of two nodes by a set of springs located at the contact facet of two rigid bodies, which is the mid-cross-section of the element. Displacement jumps are computed at the mid-cross-section, which are smeared out over the element length in the form of strains. The element is defined in x,y plane (see Figure 5). The element features are summarized in Table 6.

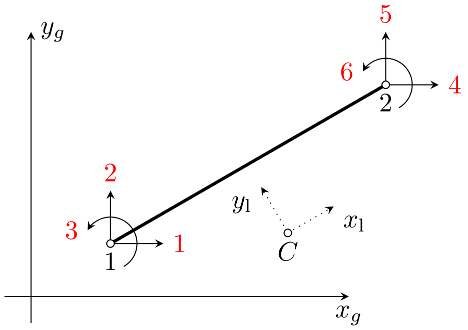

Figure 5: Lattice2d element. Node numbering, DOF numbering and definition of integration point C.

Keyword	**lattice2d**
Description	2d lattice element
Specific parameters	`thick` $\#_{(\mathrm{rn})}$ `width` $\#_{(\mathrm{rn})}$ `gpCoords` $\#_{(\mathrm{ra})}$
Parameters	`thick`: defines the out of plane (z-direction) thickness `width`: defines the width of the midpoint cross-section in the x-y plane with the point C at its centre `gpCooords`: array of the coordinates of the integration point C in the global coordinate system
Unknowns	Three dofs (u-displacement, v-displacement, w-rotation) are required in each node.

Table 6: lattice2d element summary

The theory of lattice2d is described in the paper "P. Grassl and M. Jirásek. Meso-scale approach to modelling the fracture process zone of concrete subjected to uniaxial tension. International Journal of Solids and Structures. Volume 47, Issues 7-8, pp. 957-968, 2010."

2.3.2 Lattice2dBoundary element

Represents three-node lattice element for boundary of 2d periodic cells. The first two nodes have 3 degrees of freedom as for the element lattice2d. The third node is used to control the loading of the periodic cell. It has three components which are displacements, which are produces of the macroscopic (average) strain components and length of the peridodic cell as aE_{xx}, bE_{xx} and bG_{xy} and the length of the periodic cell. The DOFs of the node that lies outside the periodic are computed from those of the periodic image inside the cell and the DOFs at the third node (Figure 6). The coordinates x and y of the third node are the lengths a and b of the periodic cell, respectively. The element is defined in x,y plane. The strain components at the additional node have the meaning of average strains in the periodic cell. The specific input parameters for this element in addition to those used for lattice2d are shown in Table 7.

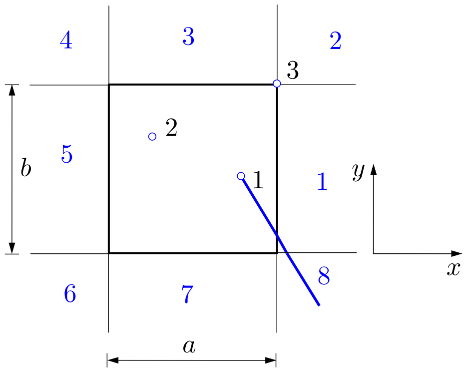

Figure 6: Lattice2dboundary element. Elements with cross boundary of periodic cell use DOFs of node inside cell and average strain values of periodic cell.

Keyword	**lattice2dboundary**
Description	2d lattice boundary element
Specific parameters	`location` $\#_{\rm (in)}$
Parameters	`location`: number between 1 and 8 which specifies the location of the node with respect to the periodic cell.
Unknowns	Nine DOFs are required, which are u-displacement, v-displacement and w-rotation at nodes 1 and 2, and aE_{xx}, bE_{yy} and bG_{xy} at node 3.
Reference	[4]

Table 7: lattice2dboundary element summary

2.3.3 Lattice3d element

Lattice3d represents a two-node 3d lattice element for small rotations. Each node has six degrees of freedom as shown in Figure 7. The element is based on the Rigid Body Spring Model originally developed by Kawai and later delveloped by Bolander for modelling fracture in concrete. The main idea is to model the elastic and inelastic response of a connection of two nodes by a set of springs located at the contact facet of two rigid bodies, which is the mid-cross-section of the element. The properties of the mid-cross-section are internally computed from its vertices which are given as input in the global coordinate sytem. Displacement jumps are computed at the mid-cross-section, which are smeared out over the element length in the form of strains. The input parameters for this element are shown in Table 8.

2.3.4 Lattice3dBoundary element

This element represents a three-noded 3d lattice element for boundaries of 3d periodic cells. The first two nodes have 6 degrees of freedom as for the element lattice3d. The third node is used to control the loading of the periodic cell using three normal (aE_{xx}, bE_{yy} and zE_{zz}) and three shear strain (cG_{yz}, cG_{xz}, bG_{xy}) components. Here, a, b and c are the three dimensions of the periodic cell. The DOFs of the node that lies outside the periodic are computed from those of the periodic image inside the cell and the DOFs at the third node (Figure 8). The connection between periodic nodes is defined as

$$\mathbf{x}^{'} = \mathbf{M}\mathbf{x} \tag{1}$$

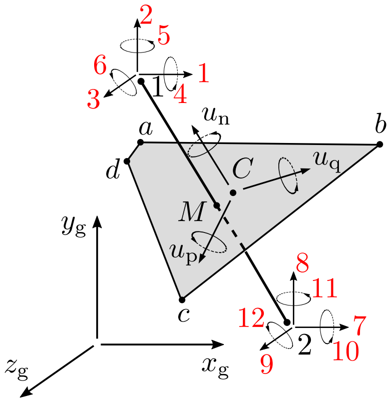

Figure 7: Lattice3d element. Node numbering, DOF numbering, cross-section vertices and local coordinate system at integration point C.

Keyword	**lattice3d**
Description	3d lattice element
Specific parameters	`polycoords` #(ra) `couplingflag` #(in) `couplingnumbers` #(ra) `pressures` #(ra) `mlength` #(rn)
Parameters	`polycooords`: array of the coordinates of the vertices of the mid-cross-section of the lattice element in the global coordinate system. `couplingflag`: flag (optional parameter. Default is 0) which activates coupling with a transport lattice element. `couplingnumbers`: array of numbers of transport lattice elements (optional parameter), which are coupled with the 3d lattice element. `pressures`: array of pressure values (optional parameter), which are used to consider influence of fluid pressure on mechanical response. `mlength`: minimum length (optional parameter) is used to check if the cross-section of the element is not too small. Default value is 1.e-20.
Unknowns	Six dofs (u-displacement, v-displacement, w-displacement, u-rotation, v-rotation and w-rotation) are required in each node.
Reference	[5]

Table 8: lattice3d element summary

Here, $\mathbf{x}^{'}$ and $\mathbf{x}$ are the nodes inside and outside, respectively, and $\mathbf{M}$ is the translation matrix, for which the input is provided in the form of a location parameter as shown in Table 9.

2.3.5 Latticelink3d element

This element represents a two-node 3d link element connecting 3d beam and 3d lattice elements. Each node has six degrees of freedom. The input parameters for this element are shown in Table 51.

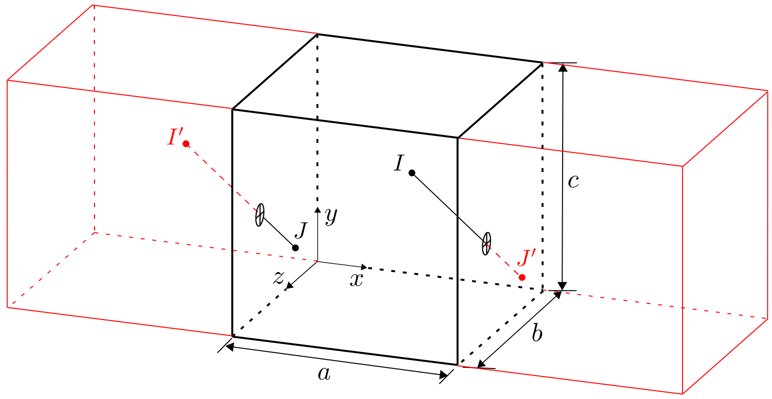

Figure 8: Lattice3dboundary element. Elements with cross boundary of periodic cell use DOFs of node inside cell and average strain values of periodic cell.

Keyword	**lattice3dboundary**
Description	3d lattice boundary element
Specific parameters	`location` $\#_{\text{(in)}}$
Parameters	`location`: array of two numbers between number between 1 and 26, which specifies the location of the two nodes with respect to the 3d periodic cell.
Unknowns	Six dofs (u-displacement, v-displacement, w-displacement, u-rotation, v-rotation and w-rotation) are required in each of the first two node. Node 3 requires the 6 quantities to control the periodic cell aE_{xx}, bE_{yy}, zE_{zz}, cG_{yz}, cG_{xz} and bG_{xy}
Reference	[6]

Table 9: lattice3dboundary element summary

Keyword	**latticelink3d**
Description	3d lattice link element
Specific parameters	`length` $\#_{\text{(rn)}}$ `diameter` $\#_{\text{(rn)}}$ `dirvector` $\#_{\text{(ra)}}$ `l_end` $\#_{\text{(rn)}}$
Parameters	`length`: bond length `diameter`: diameter `dirvector`: direction vector in which bond-slip occurs. `l_end`: array of pressure values (optional parameter), which are used to consider influence of fluid pressure on mechanical response. `mlength`: minimum length (optional parameter) is used to check if the cross-section of the element is not too small. Default value is 1.e-20.
Unknowns	Six dofs (u-displacement, v-displacement, w-displacement, u-rotation, v-rotation and w-rotation) are required in each node.
Reference	[7]

Table 10: latticelink3d element summary

2.3.6 Latticelink3dboundary element

Represents three-node 3d boundary link element connecting 3d beam and 3d lattice elements. The first two nodes have the same meaning as for latticelink3d. The third node is used to control the loading of the periodic cell using three normal (xx, yy and zz) and three shear strain (yz, xz, xy) components. The specific input parameters for this element in addition of those for latticelink3d are shown in Table 11.

Keyword	**latticelink3dboundary**
Description	3d lattice link boundary element
Specific parameters	`location` $\#_{\text{(in)}}$ rn
Parameters	`location`:
Unknowns	Six dofs (u-displacement, v-displacement, w-displacement, u-rotation, v-rotation and w-rotation) are required in each node.
Reference	[7]

Table 11: latticelink3dboundary element summary

2.3.7 Latticeframe3d element

Latticeframe3d represents a two-node linear 3d frame element shown in Figure 9. Each node has six degrees of freedom, namely three translations and three rotations. The element is based on the link between rigid body spring and Timoshenko frame models. The main idea is to model the elastic and inelastic response of a connection of two nodes by a set of springs located at the contact facet of two rigid bodies. Displacement jumps are computed at the contact facet, which are smeared out over the element length in the form of strains. The input parameters for this element are shown in Table 12. For the computation of the nodal forces, the initial configuration is used. The position of the plastic hinge is controlled by parameter s, which is normalised by half of the element length from the centre of the element. This approach was developed by [8].

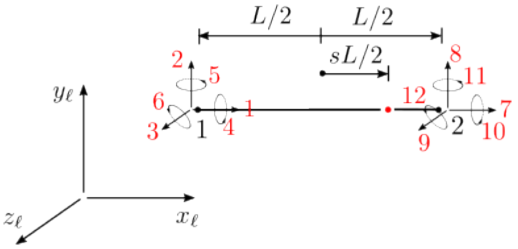

Figure 9: latticeframe3d element. Node numbering, DOF numbering and location of contact with springs.

2.3.8 Latticeframe3dnl element

Latticeframe3dnl represents a two-node geometrically nonlinear 3d frame element. This element is an extension of latticeframe3d. Each node has six degrees of freedom, namely three translations and three rotations. The element is based on the link between rigid body spring and Timoshenko frame models. The main idea is to model the elastic and inelastic response of a connection of two nodes by a set of springs located at the contact facet of two rigid bodies. Displacement jumps are computed at the contact facet using the deformed geometry assuming large deformations, which are smeared out over the element length in the form of strains. The input parameters for this element are shown in Table 13. For the computation of the nodal forces, the deformed configuration is used as illustrated in Figure 10.

Keyword	**latticeframe3d**
Description	3d lattice frame element
Specific parameters	`zaxis` $\#_{\text{(ra)}}$ [`s` $\#_{\text{(rn)}}$]
Parameters	`zaxis`: z-axis of local coordinate system `s`: distance from centre to location of springs normalised by half of element length (0 centre of element (default), -1 left node of element, 1 right end of element)
CS properties	Area, inertia moment along y and z axis (`iy` and `iz` parameters), torsion inertia moment (`ik` parameter) and either cross section area shear correction factor (`beamshearcoeff` parameter) or equivalent shear areas (`sheareareay` and `shearareaz` parameters) are required. These cross section properties are assumed to be defined in local coordinate system of element.
Unknowns	Six dofs (u-displacement, v-displacement, w-displacement, u-rotation, v-rotation and w-rotation) are required in each node.
Reference	[8]

Table 12: latticeframe3d element summary

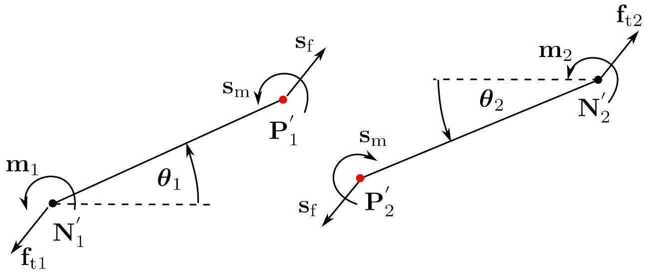

Figure 10: latticeframe3dnl element. Illustration of calculation of nodal forces using deformed geometry.

Keyword	**latticeframe3dnl**
Description	3d lattice frame element
Specific parameters	-
Parameters	
CS properties	Area, inertia moment along y and z axis (`iy` and `iz` parameters), torsion inertia moment (`ik` parameter) and either cross section area shear correction factor (`beamshearcoeff` parameter) or equivalent shear areas (`sheareareay` and `shearareaz` parameters) are required. These cross section properties are assumed to be defined in local coordinate system of element.
Unknowns	Six dofs (u-displacement, v-displacement, w-displacement, u-rotation, v-rotation and w-rotation) are required in each node.
Reference	[9]

Table 13: latticeframe3dnl element summary

2.4 Plane Stress Elements

2.4.1 PlaneStress2d

Represents isoparametric four-node quadrilateral plane-stress finite element. Each node has 2 degrees of freedom. Structure should be defined in x,y plane. The nodes should be numbered anti-clockwise (positive rotation around z-axis). The element features are summarized in Table 14.

The generalization of this element, that can be positioned arbitrarily in space is `linquad3dplanestress` element. This element requires 3 displacement degrees of freedon in each node and assumes, that the element geometry is flat, i.e. all nodes are in the same plane. The element features are summarized in Table 15.

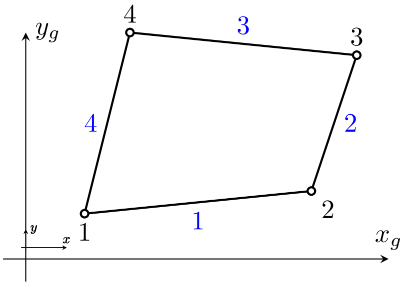

Figure 11: PlaneStress2d element. Node numbering, edge numbering and definition of local edge c.s.(a).

Keyword	**planestress2d**
Description	2D quadrilateral element for plane stress analysis
Specific parameters	[`NIP` #$_{(in)}$]
Parameters	`NIP`: allows to set the number of integration points
Unknowns	Two dofs (u-displacement, v-displacement) are required in each node.
Approximation	Linear approximation of displacements and geometry.
Integration	Integration of membrane strain terms using Gauss integration formula in 1, 4 (default), 9 or 16 integration points. The default number of integration points used can be overloaded using `NIP` parameter. Reduced integration for shear terms is employed. Shear terms are always integrated using the 1-point integration rule.
Features	Nonlocal constitutive support, Geometric nonlinearity support.
CS properties	cross section thickness is required.
Loads	Body loads are supported. Boundary loads are supported and computed using numerical integration. The side numbering is following. Each i-th element side begins in i-th element node and ends on next element node (i+1-th node or 1-st node, in the case of side number 4). The local positive edge x-axis coincides with side direction, the positive local edge y-axis is rotated 90 degrees anti-clockwise (see fig. (11)).
Nlgeo	0, 1.
Status	Reliable

Table 14: planestress2d element summary

Keyword	**linquad3dplanestress**
Description	3D quadrilateral element for plane stress analysis
Specific parameters	[`NIP` $\#_{(in)}$]
Parameters	`NIP`: allows to set the number of integration points
Unknowns	Three dofs (u-displacement, v-displacement, w-displacement) are required in each node.
Approximation	Linear approximation of displacements and geometry.
Integration	Integration of membrane strain terms using Gauss integration formula in 1, 4 (default), 9 or 16 integration points. The default number of integration points used can be overloaded using `NIP` parameter. Reduced integration for shear terms is employed. Shear terms are always integrated using the 1-point integration rule.
Features	Nonlocal constitutive support, Geometric nonlinearity support.
CS properties	cross section thickness is required.
Loads	Body loads are supported. Boundary loads are supported and computed using numerical integration. The side numbering is following. Each i-th element side begins in i-th element node and ends on next element node (i+1-th node or 1-st node, in the case of side number 4). The local positive edge x-axis coincides with side direction, the positive local edge y-axis is rotated 90 degrees anti-clockwise (see fig. (11)).
Nlgeo	0, 1.
Status	Basic functionality tested, element loads need further testing.

Table 15: linquad3dplanestress element summary

2.4.2 QPlaneStress2d

Implementation of quadratic isoparametric eight-node quadrilateral plane-stress finite element. Each node has 2 degrees of freedom. The node numbering is anti-clockwise and is explained in fig. (12). The element features are summarized in Table 16.

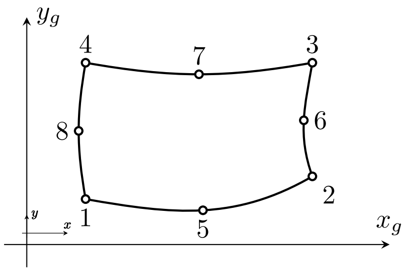

Figure 12: QPlaneStress2d element - node numbering.

2.4.3 TrPlaneStress2d

Implements an triangular three-node constant strain plane-stress finite element. Each node has 2 degrees of freedom. The node numbering is anti-clockwise. The element features are summarized in Table 17.

Keyword	**qplanestress2d**
Description	2D quadratic isoparametric plane stress element
Specific parameters	[`NIP` #(in)]
Parameters	`NIP`: allows to set the number of integration points
Unknowns	Two dofs (u-displacement, v-displacement) are required in each node.
Approximation	Quadratic approximation of displacements and geometry.
Integration	Full integration using Gauss integration formula in 4 (the default), 9 or 16 integration points. The default number of integration points used can be overloaded using `NIP` parameter.
Features	Adaptivity support.
CS properties	Cross section thickness is required.
Loads	Body and boundary loads are supported.
Nlgeo	0, 1.
Status	Stable

Table 16: qplanestress2d element summary

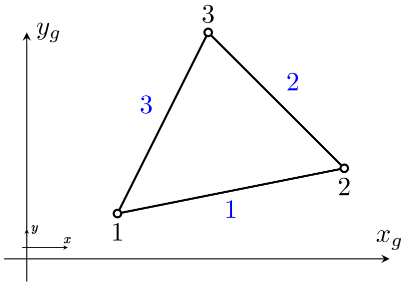

Figure 13: TrPlaneStress2d element - node and side numbering.

2.4.4 QTrPlStr

Implementation of quadratic six-node plane-stress finite element. Each node has 2 degrees of freedom. Node numbering is anti-clockwise and is shown in fig. (14). The element features are summarized in Table 18.

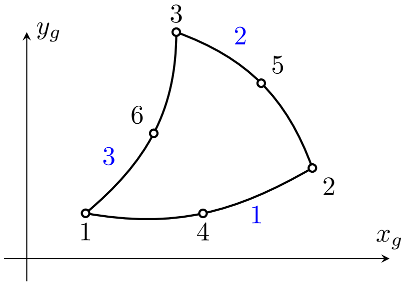

Figure 14: QTrPlStr element - node and side numbering.

2.4.5 TrPlaneStrRot

Implementation of triangular three-node plane-stress finite element with independent rotation field. Each node has 3 degrees of freedom. The element features are summarized in Table 19.

Keyword	**trplanestress2d**
Description	2D linear triangular isoparametric plane stress element
Specific parameters	-
Unknowns	Two dofs (u-displacement, v-displacement) are required in each node.
Approximation	Linear approximation of displacements and geometry.
Integration	Integration of membrane strain terms using one point gauss integration formula.
Features	Nonlocal constitutive support, Edge load support, Geometric nonlinearity support, Adaptivity support.
CS properties	Cross section thickness is required.
Loads	Body loads are supported. Boundary loads are supported and are computed using numerical integration. The side numbering is following. Each i-th element side begins in i-th element node and ends on next element node (i+1-th node or 1-st node, in the case of side number 3). The local positive edge x-axis coincides with side direction, the positive local edge y-axis is rotated 90 degrees anti-clockwise (see fig. (13)).
Nlgeo	0, 1.
Status	Reliable

Table 17: trplanestress2d element summary

Keyword	**qtrplstr**
Description	2D quadratic triangular plane stress element
Specific parameters	[`NIP` $\#_{\rm (in)}$]
Parameters	`NIP`: allows to set the number of integration points
Unknowns	Two dofs (u-displacement, v-displacement) are required in each node.
Approximation	Quadratic approximation of displacements and geometry.
Integration	Full integration using gauss integration formula in 4 points (the default) or in 7 points (using `NIP` parameter).
Features	Adaptivity support (error indicator).
CS properties	Cross section thickness is required.
Loads	Boundary loads are supported.
Nlgeo	0, 1.
Status	-

Table 18: qtrplstr element summary

The generalization of this element, that can be positioned arbitrarily in space is `trplanestrrot3d` element. This element requires 6 degrees of freedon in each node. The element features are summarized in Table 20.

The implementation is based on the following paper: Ibrahimbegovic, A., Taylor, R.L., Wilson, E. L.: A robust membrane qudritelar element with rotational degrees of freedom, Int. J. Num. Meth. Engng., 30, 445-457, 1990. The rotation field is defined as $\omega = \frac{1}{2}(\frac{\mathrm{d}v}{\mathrm{d}x} - \frac{\mathrm{d}u}{\mathrm{d}y}) = \nabla_u \boldsymbol{u}$. The following form of potential energy functial is assumed:

$$\Pi = \frac{1}{2}\int_\Omega \boldsymbol{\sigma}^T \boldsymbol{\varepsilon}\ d\Omega + \int_\Omega \boldsymbol{\tau}^T(\nabla_u \boldsymbol{u} - \omega)\ d\Omega - \int_\Omega \boldsymbol{X}^T \boldsymbol{u}\ d\Omega$$

where $\boldsymbol{\tau}$ is pseudo-stress (component of anti-symmetric stress tensor) working on dislocation $(\nabla_u \boldsymbol{u} - \omega)$; the following constitutive relation foris assumed: $\boldsymbol{\tau} = G(\nabla_u \boldsymbol{u} - \omega)$, where G is elasticity modulus in shear.

Keyword	**trplanestrrot**
Description	2D linear triangular plane stress element with rotational DOFs
Specific parameters	[`NIP` #$_{\text{(in)}}$] [`NIPRot` #$_{\text{(in)}}$]
Parameters	`NIP`: allows to set the number of integration points for integration of membrane terms. `NIPRot`: allows to set the number of integration points for integration of terms associated to rotational field.
Unknowns	Three dofs (u-displacement, v-displacement, z-rotation) are required in each node.
Approximation	Linear approximation of displacements and geometry.
Integration	Integration of membrane strain terms using gauss integration formula in 4 points (default) or using 1 or 7 points (using `NIP` parameter). Integration of strains associated with rotational field integration using 1 point is default (4 and 7 points rules can be specified using `NIPRot` parameter).
Features	-
CS properties	Cross section thickness is required.
Loads	-
Nlgeo	0.
Status	-

Table 19: trplanestrrot element summary

Keyword	**trplanestrrot3d**
Description	3D linear triangular plane stress element with rotational DOFs
Specific parameters	[`NIP` #$_{\text{(in)}}$] [`NIPRot` #$_{\text{(in)}}$]
Parameters	`NIP`: allows to set the number of integration points for integration of membrane terms. `NIPRot`: allows to set the number of integration points for integration of terms associated to rotational field.
Unknowns	Six dofs (u-displacement, v-displacement, w-displacement, x-rotation, y-rotation, z-rotation) are required in each node.
Approximation	Linear approximation of displacements and geometry.
Integration	Integration of membrane strain terms using gauss integration formula in 4 points (default) or using 1 or 7 points (using `NIP` parameter). Integration of strains associated with rotational field integration using 1 point is default (4 and 7 points rules can be specified using `NIPRot` parameter).
Features	-
CS properties	Cross section thickness is required.
Loads	-
Nlgeo	0.
Status	-

Table 20: trplanestrrot3d element summary

2.4.6 TrPlaneStressRotAllman

Implementation of triangular three-node plane-stress with nodal rotations. Each node has 3 degrees of freedom. The element features are summarized in Table 21.

The generalization of this element, that can be positioned arbitrarily in space is `trplanestressrotallman3d` element. This element requires 6 degrees of freedon in each node. The element features are summarized in Table 22.

The implementation is based on the following paper: Allman, D.J.: A compatible triangular element including vertex rotations for plane elasticity analysis, Computers & Structures, vol. 19, no. 1-2, pp. 1-8, 1984. The element is based on plane stress element with quadratic interpolation. The displacements in midside nodes are expressed using vertex displacements and vertex rotations (for edge normal displacement component); the tangential component is interpolated from vertex values. For particular element side starting at i-th vertex and ending in j-th vertex the normal and tangential displacements at edge midpoint can be expressed as

$$\begin{aligned} u_n|_{l/2} &= \frac{u_{ni}+u_{nj}}{2} + \frac{l}{8}(\omega_i - \omega_j) \\ u_t|_{l/2} &= \frac{u_{ti}+u_{tj}}{2} \end{aligned}$$

where l is edge length. This allows to express global displacements in element midside nodes using vertex displacements and rotations. For a single edge, one obtains:

$$\begin{aligned} u|_{l/2} &= -\frac{u_{ni}+u_{nj}}{2} + \frac{l}{8}(\omega_i - \omega_j)\frac{\Delta y_{ji}}{l} + (\frac{u_{t1}+u_{t2}}{2})\frac{\Delta x_{ji}}{l} \\ v|_{l/2} &= \frac{u_{ni}+u_{nj}}{2} + \frac{l}{8}(\omega_i - \omega_j)\frac{\Delta x_{ji}}{l} + (\frac{u_{t1}+u_{t2}}{2})\frac{\Delta y_{ji}}{l} \end{aligned}$$

Keyword	**trplanestressrotallman**
Description	2D linear triangular plane stress element with rotational DOFs
Specific parameters	
Unknowns	Three dofs (u-displacement, v-displacement, z-rotation) are required in each node.
Approximation	Linear approximation of geometry, quadratic interpolation of displacements.
Integration	Integration of membrane strain terms using gauss integration formula in 4 points.
Zero energy mode	The zero energy mode (equal rotations) is handled by adding additional energy term preventing spurious modes.
Features	-
CS properties	Cross section thickness is required.
Loads	-
Nlgeo	0.
Status	-

Table 21: trplanestressrotallman element summary

Keyword	**trplanestressrotallman3d**
Description	2D linear triangular plane stress element with rotational DOFs
Specific parameters	
Unknowns	Six dofs (D_u, D_v, D_w, R_x, R_y, R_z) are required in each node.
Approximation	Linear approximation of geometry, quadratic interpolation of displacements.
Integration	Integration of membrane strain terms using gauss integration formula in 4 points.
Zero energy mode	The zero energy mode (equal rotations) is handled by adding additional energy term preventing spurious modes.
Features	-
CS properties	Cross section thickness is required.
Loads	-
Nlgeo	0.
Status	-

Table 22: trplanestressrotallman3d element summary

2.5 Plane Strain Elements

2.5.1 Quad1PlaneStrain

Represents isoparametric four-node quadrilateral plane-strain finite element. Each node has 2 degrees of freedom. Structure should be defined in x,y plane. The nodes should be numbered anti-clockwise (positive rotation around z-axis). The element features are summarized in Table 23.

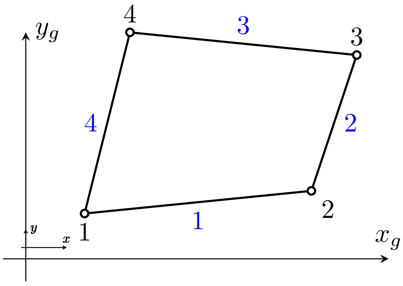

Figure 15: Quad1PlaneStrain element. Node numbering, Side numbering and definition of local edge c.s.(a).

Keyword	**quad1planestrain**
Description	2D linear quadrilateral plane-strain element
Specific parameters	[`NIP` #$_{\text{(in)}}$]
Parameters	`NIP`: allows to set the number of integration points for integration of membrane terms.
Unknowns	Two dofs (u-displacement, v-displacement) are required in each node.
Approximation	Linear approximation of displacements and geometry.
Integration	Integration of membrane strain terms using gauss integration formula in 4 (the default), 9 or 16 integration points. The default number of integration points used can be overloaded using `NIP` parameter. Reduced integration for shear terms is employed. Shear terms are always integrated using 1 point integration rule.
Features	Nonlocal constitutive support, Adaptivity support.
CS properties	Cross section thickness is required.
Loads	Body loads are supported. Boundary loads are supported and computed using numerical integration. The side numbering is following. Each i-th element side begins in i-th element node and ends on next element node (i+1-th node or 1-st node, in the case of side number 4). The local positive edge x-axis coincides with side direction, the positive local edge y-axis is rotated 90 degrees anti-clockwise (see fig. (15)).
Nlgeo	0.
Status	Reliable

Table 23: quad1planestrain element summary

2.5.2 TrplaneStrain

Implements an triangular three-node constant strain plane-strain finite element. Each node has 2 degrees of freedom. The node numbering is anti-clockwise. The element features are summarized in Table 24.

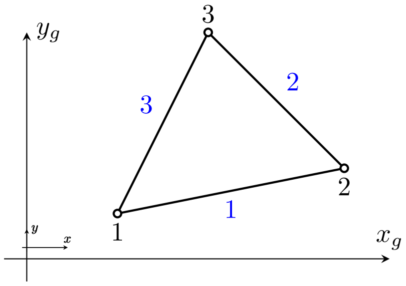

Figure 16: TrplaneStrain element - node and side numbering.

Keyword	**trplanestrain**
Description	2D linear triangular plane-strain element
Specific parameters	-
Unknowns	Two dofs (u-displacement, v-displacement) are required in each node.
Approximation	Linear approximation of displacements and geometry.
Integration	Integration of membrane strain terms using one point gauss integration formula.
Features	Nonlocal constitutive support. Edge load support, Adaptivity support.
CS properties	Cross section thickness is required.
Loads	Body loads are supported. Boundary loads are supported and are computed using numerical integration. The side numbering is following. Each i-th element side begins in i-th element node and ends on next element node (i+1-th node or 1-st node, in the case of side number 3). The local positive edge x-axis coincides with side direction, the positive local edge y-axis is rotated 90 degrees anti-clockwise (see fig. (16)).
Nlgeo	0.
Status	Reliable

Table 24: trplanestrain element summary

2.6 Plate & Shell Elements

2.6.1 DKT Element

Implementation of Discrete Kirchhoff Triangle (DKT) plate element. This element is suitable for thin plates, as the traswerse shear strain energy is neglected. The structure should be defined in x,y plane, nodes should be numbered anti-clockwise (positive rotation around z-axis). The element features are summarized in Table 25.

Keyword	**dktplate**
Description	2D Discrete Kirchhoff Triangular plate element
Specific parameters	-
Unknowns	Three dofs (w-displacement, u and v - rotations) are required in each node.
Approximation	Quadratic approximation of rotations, cubic approximation of displacement along the edges. Note: there is no need to define interpolation for displacement on the element.
Integration	Default integration of all terms using three point formula.
Features	Layered cross section support.
CS properties	Cross section thickness is required.
Loads	Body loads are supported. Boundary load support is beta.
Output	On output, the generalized shell strain/force momentum vectors in global coordinate system are printed, with the following meaning: $s_\varepsilon = \{\varepsilon_x, \varepsilon_y, \varepsilon_{xz}, \kappa_x, \kappa_y, \kappa_{xy}, \gamma_{xz}, \gamma_{yz}\},$ $s_\sigma = \{n_x, n_y, n_{xy}, m_x, m_y, m_z, m_{xy}, q_{xz}, q_{yz}\}$ where $\varepsilon_x, \varepsilon_y, \varepsilon_{xy}$ are membrane in plane normal deformations, γ_{zx}, γ_{xz} are (out of plane and in plane) shear componets, $\kappa_x, \kappa_y, \kappa_{xy}$ are curvatures, $n_x, n_y, n_{xy}, q_{xz}, q_{yz}$ are integral forces (normal and shear forces), and m_x, m_y, m_{xy} are bending moments. Please note, for example, that bending moment m_x is defined as $m_x = \int \sigma_x z \, dz$, so it acts along the y-axis and positive value causes tension in bottom layer.
Nlgeo	0.
Status	Reliable
Reference	J.L.Batoz, K.J.Bathe, L.W.Ho: A study of three-node triangular plate bending elements, IJNME, 15(12):1771-1812, 1980

Table 25: DKTplate element summary

2.6.2 QDKT Element

Implementation of Discrete Kirchhoff Theory plate quad element (QDKT). This element is suitable for thin plates, as the traswerse shear strain energy is neglected. The structure should be defined in x,y plane, nodes should be numbered anti-clockwise (positive rotation around z-axis). The element features are summarized in Table 26.

2.6.3 CCT Element

Implementation of constant curvature triangular element for plate analysis. Formulation based on Mindlin hypothesis. The structure should be defined in x,y plane. The nodes should be numbered anti-clockwise

Keyword	**qdktplate**
Description	2D Discrete Kirchhoff Quad plate element
Specific parameters	-
Unknowns	Three dofs (w-displacement, u and v - rotations) are required in each node.
Approximation	Quadratic approximation of rotations, cubic approximation of displacement along the edges. Note: there is no need to define interpolation for displacement on the element.
Integration	Default integration of all bending terms using four point formula.
Features	Layered cross section support.
CS properties	Cross section thickness is required.
Loads	Body loads are supported.
Output	On output, the generalized shell strain/force momentum vectors in global coordinate system are printed, with the following meaning: $s_\varepsilon = \{\varepsilon_x, \varepsilon_y, \varepsilon_{xz}, \kappa_x, \kappa_y, \kappa_{xy}, \gamma_{xz}, \gamma_{yz}\},$ $s_\sigma = \{n_x, n_y, n_{xy}, m_x, m_y, m_z, m_{xy}, q_{xz}, q_{yz}\}$ where $\varepsilon_x, \varepsilon_y, \varepsilon_{xy}$ are membrane in plane normal deformations, γ_{zx}, γ_{xz} are (out of plane and in plane) shear componets, $\kappa_x, \kappa_y, \kappa_{xy}$ are curvatures, $n_x, n_y, n_{xy}, q_{xz}, q_{yz}$ are integral forces (normal and shear forces), and m_x, m_y, m_{xy} are bending moments. Please note, for example, that bending moment m_x is defined as $m_x = \int \sigma_x z \; dz$, so it acts along the y-axis and positive value causes tension in bottom layer.
Nlgeo	0.
Status	Reliable
Reference	J.L.Batoz, K.J.Bathe, L.W.Ho: A study of three-node triangular plate bending elements, IJNME, 15(12):1771-1812, 1980

Table 26: QDKTplate element summary

(positive rotation around z-axis). The element features are summarized in Table 27.

2.6.4 CCT3D Element

Implementation of constant curvature triangular element for plate analysis. Formulation based on Mindlin hypothesis. The element could be arbitrarily oriented in space. The nodes should be numbered anti-clockwise (positive rotation around element normal). The element features are summarized in Table 28.

2.6.5 RerShell Element

Combination of CCT plate element (Mindlin hypothesis) with triangular plane stress element for membrane behavior. The element curvature can be specified. Although element requires generally six DOFs per node, no stiffness to local rotation along z-axis (rotation around element normal) is supplied. The element features are summarized in Table 29.

Keyword	**cctplate**
Description	2D constant curvature triangular plate element
Specific parameters	-
Unknowns	Three dofs (w-displacement, u and v - rotations) are required in each node.
Approximation	Linear approximation of rotations, quadratic approximation of displacement.
Integration	Integration of all terms using one point formula.
Features	Layered cross section support.
CS properties	Cross section thickness is required.
Loads	Body loads are supported. Boundary loads are not supported now.
Output	On output, the generalized shell strain/force momentum vectors in global coordinate system are printed, with the following meaning: $s_\varepsilon = \{\varepsilon_x, \varepsilon_y, \varepsilon_{xz}, \kappa_x, \kappa_y, \kappa_{xy}, \gamma_{xz}, \gamma_{yz}\},$ $s_\sigma = \{n_x, n_y, n_{xy}, m_x, m_y, m_z, m_{xy}, q_{xz}, q_{yz}\}$ where $\varepsilon_x, \varepsilon_y, \varepsilon_{xy}$ are membrane in plane normal deformations, γ_{zx}, γ_{xz} are (out of plane and in plane) shear componets, $\kappa_x, \kappa_y, \kappa_{xy}$ are curvatures, $n_x, n_y, n_{xy}, q_{xz}, q_{yz}$ are integral forces (normal and shear forces), and m_x, m_y, m_{xy} are bending moments. Please note, for example, that bending moment m_x is defined as $m_x = \int \sigma_x z\, dz$, so it acts along the y-axis and positive value causes tension in bottom layer.
Nlgeo	0.
Status	Reliable

Table 27: cctplate element summary

2.6.6 tr_shell11 element

Combination of CCT3D plate element (Mindlin hypothesis) with triangular plane stress element for membrane behavior. It comes with complete set of 6 DOFs per node. The element features are summarized in Table 30.

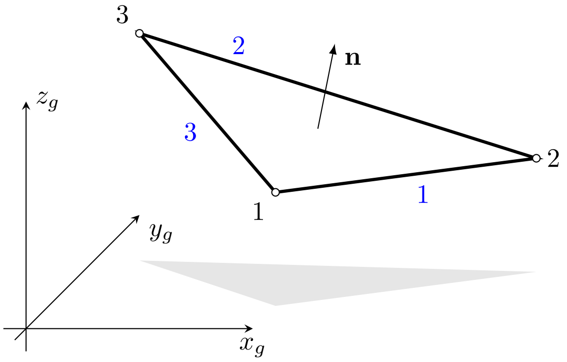

Figure 17: Geometry of tr_shell11 element.

Keyword	**cctplate3d**
Description	Constant curvature triangular plate element in arbitray position
Specific parameters	-
Unknowns	Six dofs (u,v,w-displacements and u,v,w rotations) are in general required in each node.
Approximation	Linear approximation of ratations, quadratic approximation of displacement.
Integration	Integration of all terms using one point formula.
Features	Layered cross section support.
CS properties	Cross section thickness is required.
Loads	Body loads are supported. Boundary loads are not supported now.
Output	On output, the shell force (s_f), shell strain (s_s), shell momentum (s_m), and shell curvature (s_c) tensors in global coordinate system are printed as vector form with 6 components, with the following meaning: $s_f = \{n_x, n_y, n_z, v_{yz}, v_{xz}, v_{xy}\}$, $s_s = \{\varepsilon_x, \varepsilon_y, \varepsilon_z, \gamma_{yz}, \gamma_{xz}, \gamma_{xy}\}$, $s_m = \{m_x, m_y, m_z, m_{yz}, m_{xz}, m_{xy}\}$, $s_c = \{\kappa_x, \kappa_y, \kappa_z, \kappa_{yz}, \kappa_{xz}, \kappa_{xy}\}$ where $\varepsilon_x, \varepsilon_y, \varepsilon_z$ are membrane normal deformations, $\gamma_{zy}, \gamma_{zx}, \gamma_{xy}$ are (out of plane and in plane) shear componets, $\kappa_x, \kappa_y, \kappa_z, \kappa_{yz}, \kappa_{xz}, \kappa_{xy}$ are curvatures, $n_x, n_y, n_z, v_{yz}, v_{xz}, v_{xy}$ are integral forces (normal and shear forces), and $m_x, m_y, m_z, m_{yz}, m_{xz}, m_{xy}$ are bending moments. Please note, for example, that bending moment m_x is defined as $m_x = \int \sigma_x z \, dz$, so it acts along the y-axis and positive value causes tension in bottom layer.
Nlgeo	0.
Status	Reliable

Table 28: cctplate3d element summary

2.6.7 tr_shell02 element

Combination of thin-plate DKT plate element with plane stress element (TrPlanestressRotAllman). This element comes with complete set of 6 DOFs per node. The element features are summarized in Table 31.

2.6.8 Quad1Mindlin Element

This class implements an quadrilateral, bilinear, four-node Mindlin plate. This type of element exhibit strong shear locking (thin plates exhibit almost no bending). Implements the lumped mass matrix. The element features are summarized in Table 32.

2.6.9 Tr2Shell7 Element

This class implements a triangular, quadratic, six-node shell element. The element is a so-called seven parameter shell with seven dofs per node – a displacement field (3 dofs), an extensible director field (3 dofs) and a seventh

Keyword	**rershell**
Description	Simple shell based on combination of CCT plate element (Mindlin hypothesis) with triangular plane stress element. element can be arbitrary positioned in space.
Specific parameters	-
Unknowns	Six dofs (u,v,w-displacements and u,v,w rotations) are in general required in each node. Note, that although element it requires generally six DOFs per node, no stiffness to local rotation along z-axis (rotation around element normal) is supplied.
Approximation	Linear approximation of ratations, quadratic approximation of displacement.
Integration	Integration of all terms using one point formula.
Features	Layered cross section support.
CS properties	Cross section thickness is required.
Loads	Body loads are supported. Boundary loads are not supported now.
Output	On output, the shell force (s_f), shell strain (s_s), shell momentum (s_m), and shell curvature (s_c) tensors in **global coordinate system** are printed as vector form with 6 components, with the following meaning: $$\begin{aligned} s_f &= \{n_x, n_y, n_z, v_{yz}, v_{xz}, v_{xy}\}, \\ s_s &= \{\varepsilon_x, \varepsilon_y, \varepsilon_z, \gamma_{yz}, \gamma_{xz}, \gamma_{xy}\}, \\ s_m &= \{m_x, m_y, m_z, m_{yz}, m_{xz}, m_{xy}\}, \\ s_c &= \{\kappa_x, \kappa_y, \kappa_z, \kappa_{yz}, \kappa_{xz}, \kappa_{xy}\} \end{aligned}$$ where $\varepsilon_x, \varepsilon_y, \varepsilon_z$ are membrane normal deformations, $\gamma_{zy}, \gamma_{zx}, \gamma_{xy}$ are (out of plane and in plane) shear componets, $\kappa_x, \kappa_y, \kappa_z, \kappa_{yz}, \kappa_{xz}, \kappa_{xy}$ are curvatures, $n_x, n_y, n_z, v_{yz}, v_{xz}, v_{xy}$ are integral forces (normal and shear forces), and $m_x, m_y, m_z, m_{yz}, m_{xz}, m_{xy}$ are bending moments. Please note, for example, that bending moment m_x is defined as $m_x = \int \sigma_x z \; dz$, so it acts along the y-axis and positive value causes tension in bottom layer.
Nlgeo	0.
Status	Reliable

Table 29: rershell element summary

dof representing inhomogenous thickness strain. This last parameter is included in the model in order to deal with volumetric/Poisson lock effects.

The element features are summarized in Table 32.

2.6.10 MITC4Shell Element

A four-node quadrilateral shell element formulated using three-dimensional continuum mechanics theory degenerated to shell behaviour. The element is applicable to thick and thin shells as the “mixed interpolation of tensorial components” (MITC) approach is used to remove shear locking. The implementation is based on the following paper: Dvorkin, E.N., Bathe, K.J.: A continuum mechanics based four-node shell element for general non-linear analysis, Eng.Comput., Vol.1, 77-88, 1984.

Although element requires generally six DOFs per node, no stiffness to local rotation along z-axis (rotation around director vector) is supplied. The element features are summarized in Table 34.

Keyword	**tr_shell11**
Description	Triangular shell element combining CCT3D plate element (Mindlin hypothesis) with triangular plane stress element with rotational DOFs
Specific parameters	-
Unknowns	Six dofs (u,v,w-displacements and u,v,w rotations) are in general required in each node.
Approximation	See description of cct and trplanstrrot elements
Integration	Four point integration, reduced integration of strain associated to normal rotation and shear terms.
Features	Layered cross section support.
CS properties	Cross section thickness is required.
Loads	Body loads are supported. Boundary loads are supported (only surface loads).
Output	On output, the shell force (s_f), shell strain (s_s), shell momentum (s_m), and shell curvature (s_c) tensors in **global coordinate system** are printed as vector form with 6 components, with the following meaning: $s_f = \{n_x, n_y, n_z, v_{yz}, v_{xz}, v_{xy}\},$ $s_s = \{\varepsilon_x, \varepsilon_y, \varepsilon_z, \gamma_{yz}, \gamma_{xz}, \gamma_{xy}\},$ $s_m = \{m_x, m_y, m_z, m_{yz}, m_{xz}, m_{xy}\},$ $s_c = \{\kappa_x, \kappa_y, \kappa_z, \kappa_{yz}, \kappa_{xz}, \kappa_{xy}\}$ where $\varepsilon_x, \varepsilon_y, \varepsilon_z$ are membrane normal deformations, $\gamma_{zy}, \gamma_{zx}, \gamma_{xy}$ are (out of plane and in plane) shear componets, $\kappa_x, \kappa_y, \kappa_z, \kappa_{yz}, \kappa_{xz}, \kappa_{xy}$ are curvatures, $n_x, n_y, n_z, v_{yz}, v_{xz}, v_{xy}$ are integral forces (normal and shear forces), and $m_x, m_y, m_z, m_{yz}, m_{xz}, m_{xy}$ are bending moments. Please note, for example, that bending moment m_x is defined as $m_x = \int \sigma_x z \, dz$, so it acts along the y-axis and positive value causes tension in bottom layer.
Nlgeo	0.
Status	Reliable
Tests	sm/trshell11_1.in

Table 30: tr_shell01 element summary

2.6.11 Sub-soil Elements

2.6.12 quad1plateSubsoil Element

This class implements an quadrilateral, bilinear, four-node plate subsoil element. Typically this element is combined with suitable plate element with quadrilateral geometry to model plate element on (elastic) subsoill foundation, but it can be used alone. The element geometry should be define in xy plane. The element features are summarized in Table 35.

2.6.13 Tria1PlateSubSoil Element

This class implements an quadrilateral, bilinear, four-node plate subsoil element. Typically this element is combined with suitable plate element with quadrilateral geometry to model plate element on (elastic) subsoill

Keyword	**tr_shell02**
Description	Triangular shell element combining DKT plate element with triangular plane stress element with rotational DOFs
Specific parameters	-
Unknowns	Six dofs (u,v,w-displacements and u,v,w rotations) are in general required in each node.
Approximation	See description of cct and trplanstrrot elements
Integration	4 integration points necessary, use "NIP 4" in element record.
CS properties	Cross section thickness is required.
Loads	Body loads are supported. Boundary loads are supported (only surface loads).
Output	On output, the shell force (s_f), shell strain (s_s), shell momentum (s_m), and shell curvature (s_c) tensors in **global coordinate system** are printed as vector form with 6 components, with the following meaning: $$\begin{aligned} s_f &= \{n_x, n_y, n_z, v_{yz}, v_{xz}, v_{xy}\}, \\ s_s &= \{\varepsilon_x, \varepsilon_y, \varepsilon_z, \gamma_{yz}, \gamma_{xz}, \gamma_{xy}\}, \\ s_m &= \{m_x, m_y, m_z, m_{yz}, m_{xz}, m_{xy}\}, \\ s_c &= \{\kappa_x, \kappa_y, \kappa_z, \kappa_{yz}, \kappa_{xz}, \kappa_{xy}\} \end{aligned}$$ where $\varepsilon_x, \varepsilon_y, \varepsilon_z$ are membrane normal deformations, $\gamma_{zy}, \gamma_{zx}, \gamma_{xy}$ are (out of plane and in plane) shear componets, $\kappa_x, \kappa_y, \kappa_z, \kappa_{yz}, \kappa_{xz}, \kappa_{xy}$ are curvatures, $n_x, n_y, n_z, v_{yz}, v_{xz}, v_{xy}$ are integral forces (normal and shear forces), and $m_x, m_y, m_z, m_{yz}, m_{xz}, m_{xy}$ are bending moments. Please note, for example, that bending moment m_x is defined as $m_x = \int \sigma_x z \, dz$, so it acts along the y-axis and positive value causes tension in bottom layer.
Nlgeo	0.
Status	-
Note	Works only with linear material models, as bending and membrane actions are uncoupled

Table 31: tr_shell02 element summary

foundation, but it can be used alone. The element geometry should be define in xy plane. The element features are summarized in Table 35.

Keyword	**quad1mindlin**
Description	Quadrilateral, bilinear, four-node Mindlin plate
Specific parameters	[`NIP` #$_{\text{(in)}}$]
Unknowns	Three dofs (w-displacement, u and v - rotation) are required in each node.
Approximation	Linear for all unknowns.
Integration	Default uses 4 integration points. No reduced integration is used, as it causes numerical problems.
Features	Layered cross section support.
CS properties	Cross section thickness is required.
Loads	Dead weight loads, and edge loads are supported.
Output	On output, the generalized shell strain/force momentum vectors in global coordinate system are printed, with the following meaning: $s_\varepsilon = \{\varepsilon_x, \varepsilon_y, \varepsilon_{xz}, \kappa_x, \kappa_y, \kappa_{xy}, \gamma_{xz}, \gamma_{yz}\},$ $s_\sigma = \{n_x, n_y, n_{xy}, m_x, m_y, m_z, m_{xy}, q_{xz}, q_{yz}\}$ where $\varepsilon_x, \varepsilon_y, \varepsilon_{xy}$ are membrane in plane normal deformations, γ_{zx}, γ_{xz} are (out of plane and in plane) shear componets, $\kappa_x, \kappa_y, \kappa_{xy}$ are curvatures, $n_x, n_y, n_{xy}, q_{xz}, q_{yz}$ are integral forces (normal and shear forces), and m_x, m_y, m_{xy} are bending moments. Please note, for example, that bending moment m_x is defined as $m_x = \int \sigma_x z\ dz$, so it acts along the y-axis and positive value causes tension in bottom layer.
Nlgeo	0.
Reference	[1]
Status	Experimental

Table 32: quad1mindlin element summary

Keyword	**tr2shell7**
Description	Triangular, quadratic, six-node shell with 7 dofs/node
Specific parameters	[`NIP` #$_{\text{(in)}}$]
Unknowns	Seven dofs (displacement in u, v and w-direction; change in director field in u, v and w-direction; and inhomgenous thickness stretch) are required in each node.
Approximation	Quadratic for all unknowns.
Integration	Default uses 6 integration points in the midsurface plane. Number of integration points in the thickness direction is determined by the Layered cross section.
Features	Layered cross section support.
CS properties	This element must be used with a Layered cross section.
Loads	Edge loads, constant pressure loads and surface loads are supported.
Nlgeo	Not applicable. The implementation is for large defomrations and hence geometrical nonlinearities will always be present, regardless the value of Nlgeo.
Reference	[3]
Status	Experimental

Table 33: tr2shell7 element summary

Keyword	**mitc4shell**
Description	Quadrilateral, bilinear, four-node shell element using the MITC technique.
Specific parameters	[`NIP` #$_{\mathrm{(in)}}$] [`NIPZ` #$_{\mathrm{(in)}}$] [`directorType` #$_{\mathrm{(in)}}$]
Parameters	`NIP`: allows to set the number of integration points in local x-y plane (default 4). `NIPZ`: allows to set the number of integration points in local z-direction (default 2). `directorType`: allows to set director vectors. Director vectors can be set as normal to the plane (`directorType` = 0, default), or calculated for each node as an average of neighbouring elements of same crosssection (`directorType` = 1), or can be specified at crosssection (`directorType` =2).
Unknowns	Six dofs (u,v,w-displacements and u,v,w rotations) are in general required in each node. Note, that although element requires generally six DOFs per node, no stiffness to local rotation along z-axis (rotation around director vector) is supplied.
Approximation	Linear approximation of displacements and rotations.
Integration	Integration of all terms using Gauss integration formula in 8 points (default) or specified using `NIP` and `NIPZ` parameters.
Features	Variable cross section support.
CS properties	Cross section thickness is required (measured along director vector). Director vectors components may be specified [`directorx` #$_{\mathrm{(in)}}$][`directory` #$_{\mathrm{(in)}}$][`directorz` #$_{\mathrm{(in)}}$] in case of `directorType 2`.
Loads	Body and boundary loads are supported.
Output	On output, the shell force (s_f), shell momentum (s_m), shell strain (s_s), shell curvature (s_c), strain (ε), and stress (σ) tensors in **global coordinate system** are printed as vector form with 6 components, with the following meaning: $s_f = \{n_x, n_y, n_z, v_{yz}, v_{xz}, v_{xy}\},$ $s_m = \{m_x, m_y, m_z, m_{yz}, m_{xz}, m_{xy}\},$ $s_s = \{\varepsilon_x, \varepsilon_y, \varepsilon_z, \gamma_{yz}, \gamma_{xz}, \gamma_{xy}\},$ $s_c = \{\kappa_x, \kappa_y, \kappa_z, \kappa_{yz}, \kappa_{xz}, \kappa_{xy}\}$ $\varepsilon = \{\varepsilon_x, \varepsilon_y, \varepsilon_z, \gamma_{yz}, \gamma_{zx}, \gamma_{xy}\},$ $\sigma = \{\sigma_x, \sigma_y, \sigma_z, \sigma_{yz}, \sigma_{xz}, \sigma_{xy}\}.$ where $n_x, n_y, n_z, v_{yz}, v_{xz}, v_{xy}$ are integral forces (normal and shear forces), and $m_x, m_y, m_z, m_{yz}, m_{xz}, m_{xy}$ are bending moments, $\varepsilon_x, \varepsilon_y, \varepsilon_z$ are membrane normal deformations, $\gamma_{zy}, \gamma_{zx}, \gamma_{xy}$ are (out of plane and in plane) shear componets, $\kappa_x, \kappa_y, \kappa_z, \kappa_{yz}, \kappa_{xz}, \kappa_{xy}$ are curvatures. Please note, for example, the bending moment m_x is defined as $m_x = \int \sigma_x z \, dz$, so it acts along the y-axis and positive value causes tension in bottom layer (positive z-coordinate). The shell force (s_f), shell momentum (s_m), shell strain (s_s), and shell curvature (s_c) tensors are evaluated at the midplane of the element (thus are constant along the thickness) while the strain (ε), and stress (σ) tensors are evaluated at each Gausspoint.
Nlgeo	0.
Status	-

Table 34: mitc4shell element summary

Keyword	**quad1plateSubsoil**
Description	Quadrilateral, bilinear, four-node sub-soil plate element
Specific parameters	
Unknowns	One dof (w-displacement) is required in each node.
Approximation	Linear for transwersal displacement.
Integration	4 integration points.
Loads	Surface load support.
Note	Requires material model with 2dPlateSubSoil mode support.
Reference	[2]

Table 35: quad1platesubsoil element summary

Keyword	**tria1platesubsoil**
Description	Tringular, three-node sub-soil plate element with linear interpolation
Specific parameters	
Unknowns	One dof (w-displacement) is required in each node.
Approximation	Linear for transwersal displacement.
Integration	1 integration points.
Loads	Surface load support.
Note	Requires material model with 2dPlateSubSoil mode support.
Reference	[2]

Table 36: tria1platesubsoil element summary

2.7 Axisymmetric Elements

Implementation relies on elements located exclusively in x, y plane. The coordinate x corresponds to radius, y is the axis of rotation. Approximation of displacement functions u, v is carried out on a particular finite element. Nonzero strains read

$$
\begin{aligned}
\varepsilon_x = \varepsilon_r &= \frac{\partial u}{\partial x} \qquad (2)\\
\varepsilon_y = \varepsilon_z &= \frac{\partial v}{\partial y} \qquad (3)\\
\varepsilon_\theta &= \frac{u}{r} \qquad (4)\\
\gamma_{xy} = \gamma_{rz} &= \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \qquad (5)
\end{aligned}
$$

Stress components can be computed from elasticity matrix. Note that this matrix corresponds to a submatrix of the full 3D elasticity matrix.

$$
\left\{ \begin{array}{c} \sigma_x \\ \sigma_y \\ \sigma_\theta \\ \sigma_{xy} \end{array} \right\} = \frac{E}{(1+\nu)(1-2\nu)} \left[\begin{array}{cccc} 1-\nu & \nu & \nu & 0 \\ \nu & 1-\nu & \nu & 0 \\ \nu & \nu & 1-\nu & 0 \\ 0 & 0 & 0 & (1-2\nu)/2 \end{array} \right] = \left\{ \begin{array}{c} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_\theta \\ \gamma_{xy} \end{array} \right\} \qquad (6)
$$

In OOFEM, the strain vector is arranged as $\{\varepsilon_x, \varepsilon_y, \varepsilon_\theta, 0, 0, \gamma_{xy}\}^T$ and the stress vector $\{\sigma_x, \sigma_y, \sigma_\theta, 0, 0, \tau_{xy}\}^T$. Implementation assumes a segment of 1 rad.

2.7.1 Axisymm3d element

Implementation of triangular three-node finite element for axisymmetric continuum. Each node has 2 degrees of freedom. Node numbering and edge position is the same as in Fig. 13. The element features are summarized in Table 37.

Keyword	**Axisymm3d**
Description	Triangular axisymmetric linear element
Specific parameters	[`NIP` #$_{\text{(in)}}$] [`NIPfish` #$_{\text{(in)}}$]
Parameters	`NIP`: allows to set the number of integration points (possible completions are 1 (default), 4 and 7 point integration rule).
Unknowns	Two dofs (u-displacement, v-displacement) are required in each node.
Approximation	Linear approximation of displacement and geometry.
Integration	The integration can be altered using `NIP` paramter (default is 1 point integration).
Features	-
CS properties	-
Loads	Boundary and body loads are supported.
Nlgeo	0.
Status	-

Table 37: Axisymm3d element summary

2.7.2 Q4axisymm element

Implementation of quadratic isoparametric eight-node quadrilateral - finite element for axisymmetric 3d continuum. Each node has 2 degrees of freedom. The element features are summarized in Table 38.

Keyword	**Q4axisymm**
Description	Quadratic isoparametric eight-node quadrilateral for axisymmetric analysis
Specific parameters	[`NIP` $\#_{\rm (in)}$] [`NIPfish` $\#_{\rm (in)}$]
Parameters	`NIP`: allows to set the number of integration points for integration of terms corresponding to ε_x and ε_y strains (possible completions are 1, 4 (default), 9, and 16). `NIPfish`: allows to set the number of integration points for integration of remain terms (corresponding to ε_θ and γ_{rz}) (Supported values include 1 (default), 4, 9, and 16 integration point formula).
Unknowns	Two dofs (u-displacement, v-displacement) are required in each node.
Approximation	Quadratic approximation of displacement and geometry.
Integration	The integration of terms corresponding to ε_x and ε_y strains can be altered using `NIP` parameter (default is 4 point formula). The remaining terms (creesponding to ε_θ and γ_{rz}) are integrated by default using 1 point formula (see `NIPfish` parameter).
Features	-
CS properties	-
Loads	No boundary and body loads are supported.
Nlgeo	0.
Status	-

Table 38: Q4axisymm element summary

2.7.3 L4axisymm element

Implementation of isoparametric four-node quadrilateral axisymmetric finite element with linear interpolations of displacements u, v. Node numbering and edge position is the same as in Fig. 11. The element features are summarized in Table 39.

Keyword	**L4axisymm**
Description	Isoparametric four-node quadrilateral element for axisymmetric analysis
Specific parameters	[`NIP` #$_{\rm (in)}$]
Parameters	`NIP`: allows to set the number of integration points for integration of terms corresponding to ε_x and ε_y strains (possible completions are 1, 4 (default), 9, and 16).
Unknowns	Two dofs (u-displacement, v-displacement) are required in each node.
Approximation	Linear approximation of displacement and geometry.
Integration	The integration of ε_x and ε_y strains can be altered using `NIP` parameter (possible completions are 1, 4 (default), 9 or 16 point integration rule). The remaining strain components (ε_θ and γ_{rz}) are integrated using one point integration formula.
Features	-
CS properties	-
Loads	Boundary and body loads supported.
Nlgeo	0.
Status	-

Table 39: L4axisymm element summary

2.8 3D Continuum Elements

This section contains description of continuum elements.

2.8.1 LSpace element

Implementation of Linear 3d eight - node finite element. Each node has 3 degrees of freedom. The element features are summarized in Table 40.

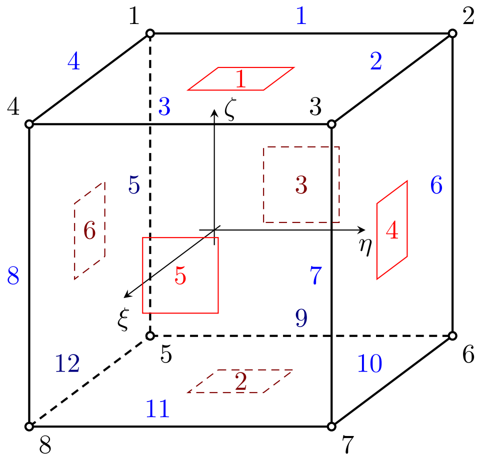

Figure 18: LSpace element (Node numbers in black, side numbers in blue, and surface numbers in red).

Keyword	**lspace**
Description	Linear isoparametric brick element
Specific parameters	[`NIP` #$_{\rm (in)}$]
Parameters	`NIP`: allows to set the number of integration points (possible completions are 1, 8 (default), or 27).
Unknowns	Three dofs (u-displacement, v-displacement, w-displacement) are required in each node.
Approximation	Linear approximation of displacement and geometry.
Integration	Full integration of all strain components.
Features	Supports adaptivity, geometric nonlinearity, and layered cross section support
CS properties	-
Loads	-
Nlgeo	0,1,2.
Status	Reliable

Table 40: lspace element summary

2.8.2 LSpaceBB element

Implementation of 3d brick eight - node linear approximation element with selective integration of deviatoric and volumetric strain contributions (B-bar formulation) for incompressible problems. Features and description identical to conventional lspace element, see section 2.8.1.

2.8.3 QSpace element

Implementation of quadratic 3d 20-node finite element. Each node has 3 degrees of freedom. The element features are summarized in Table 41.

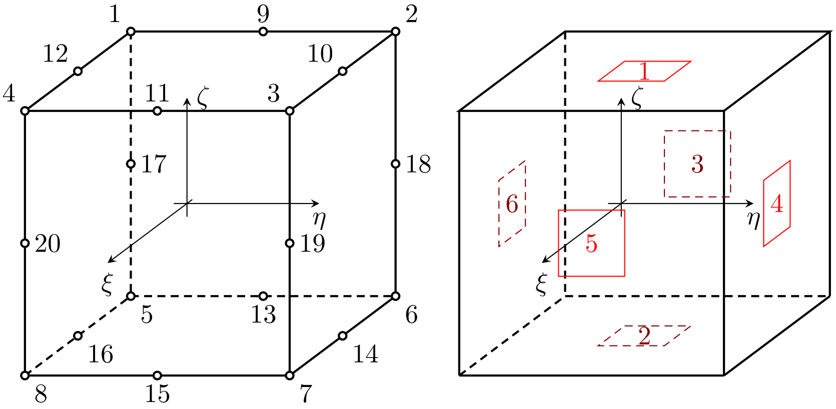

Figure 19: QSpace element.

Keyword	**qspace**
Description	Quadratic isoparametric brick element
Specific parameters	[`NIP` $\#_{\rm (in)}$]
Parameters	`NIP`: allows to set the number of integration points (possible completions are 1, 8 (default), or 27).
Unknowns	Three dofs (u-displacement, v-displacement, w-displacement) are required in each node.
Approximation	Quadratic approximation of displacement and geometry.
Integration	Full integration of all strain components.
Features	Layered cross section support.
CS properties	-
Loads	-
Nlgeo	0,1,2.
Status	Reliable

Table 41: qspace element summary

2.8.4 LTRSpace element

Implementation of tetrahedra four-node finite element. Each node has 3 degrees of freedom. The element features are summarized in Table 42. Following node numbering convention is adopted (see also Fig. 20):

- Select a face that will contain the first three corners. The excluded corner will be the last one.
- Number these three corners in a counterclockwise sense when looking at the face from the excluded corner.

2.8.5 QTRSpace element

Implementation of tetrahedra ten-node finite element. Each node has 3 degrees of freedom. The element features are summarized in Table 43. Following node numbering convention is adopted (see also Fig. 21):

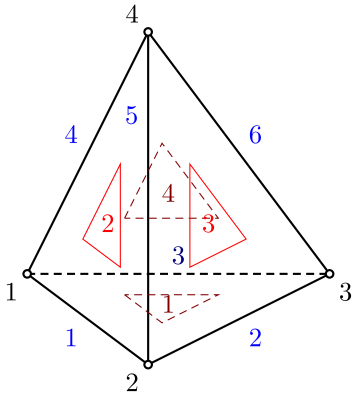

Figure 20: LTRSpace element. Definition and node numbering convention.

Keyword	**LTRSpace**
Description	Linear tetrahedra element
Specific parameters	-
Unknowns	Three dofs (u-displacement, v-displacement, w-displacement) are required in each node.
Approximation	Linear approximation of displacements and geometry using linear volume coordinates.
Integration	Full integration of all strain components using four point Gauss integration formula.
Features	Adaptivity support, Geometric nonlinearity support.
CS properties	-
Loads	Surface and Edge loadings supported.
Nlgeo	0,1,2.
Status	Reliable

Table 42: LTRSpace element summary

2.8.6 LWedge element

Implementation of wedge six-node finite element. Each node has 3 degrees of freedom. The element features are summarized in Table 44. Following node numbering convention is adopted (see also Fig. 22):

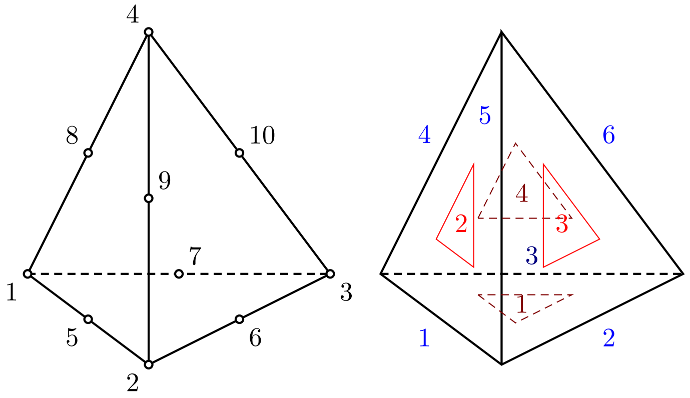

Figure 21: QTRSpace element. Definition and node numbering convention.

Keyword	**QTRSpace**
Description	3D tetrahedra element with quadratic interpolation
Specific parameters	[`NIP` #$_{\mathrm{(in)}}$]
Parameters	`NIP`: allows to alter the default integration formula (possible completions are 1, 4 (default), 5, 11, 15, 24, and 45 point intergartion formulas).
Unknowns	Three dofs (u-displacement, v-displacement, w-displacement) are required in each node.
Approximation	Quadratic approximation of displacements and geometry using linear volume coordinates.
Integration	Full integration of all strain components using four point Gauss integration formula.
Features	-
CS properties	-
Loads	-
Nlgeo	0,1,2.
Status	Reliable

Table 43: QTRSpace element summary

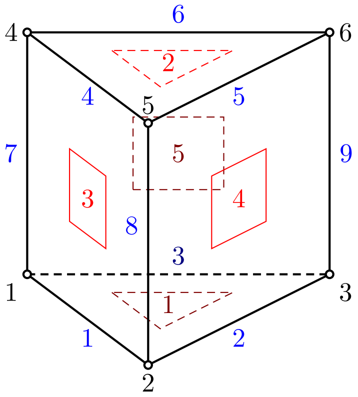

Figure 22: LWedge element. Node numbering convention in black, edge numbering in blue and face numbering in red.

2.8.7 QWedge element

Implementation of wedge fifteen-node finite element. Each node has 3 degrees of freedom. The element features are summarized in Table 45. Following node numbering convention is adopted (see also Fig. 23):

2.8.8 Layer stacking sequence definition for 3D elements

Selected 3D elements (bricks and wedge geometries) support using the LayeredCrossSection model to define layer stack as a sequence of individual layers. Individual layers are assumed to lie in element parametric $\xi - \eta$ plane and are stacked along parametric ζ coordinate of the element. The direction of parametric coordinates is determined by element node numbering convention, see figures with element geometries above. Note, that the stacking direction is in general the function of element geometry.

It is important to understand concept of element and material coordinate systems.

The element coordinate system (elemCS) coincides, by default, with the global coordinate system. The user-defined element coordinate system can be defined using lcs parameter. The lcs parameter defines an array of size 6, where the first 3 components define direction of local element-axis and remaining 3 components define direction of local element y-axis. The local element z axis is computed using vector product $e_z = e_x \times e_y$.

Keyword	**LWedge**
Description	3D wedge six-node finite element with linear interpolation
Specific parameters	[`NIP` $\#_{\rm (in)}$]
Parameters	`NIP`: allows to alter the default integration formula (possible completions are 2 (default) and 9 point integration formulas).
Unknowns	Three dofs (u-displacement, v-displacement, w-displacement) are required in each node.
Approximation	Linear approximation of displacements and geometry.
Integration	Full integration of all strain components using four point Gauss integration formula.
Features	Layered cross section support.
CS properties	-
Loads	-
Nlgeo	0,1,2.
Status	Reliable

Table 44: LWedge element summary

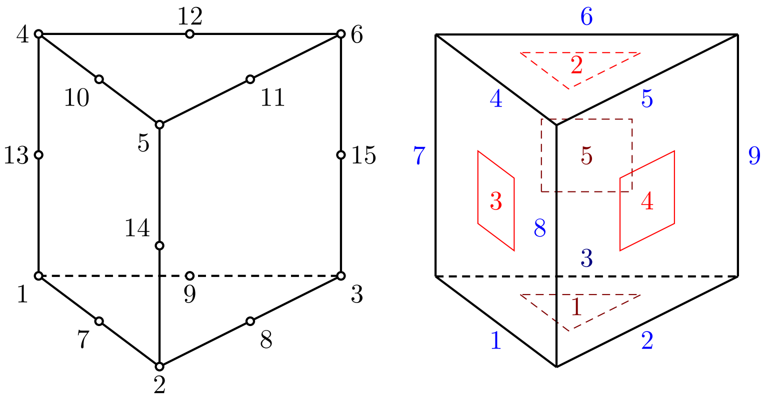

Figure 23: QWedge element. Node numbering convention in black, edge numbering in blue and face numbering in red.

The material properties of each layer are defined in material coordinate system (matCS). Also the solver output for individual layers is done in matCS. By default, the material coordinate system coincides with global coordinate system. Additionally, the material coordinate system for individual layer can be rotated around material CS z-axis by angle, defined by layered cross section `rotations` keyword. This array parameter allows to define rotation angle for individual layers and should be defined in degrees not radians. If `matcs` element keyword is present, but no lcs element record is defined, then the following definition of elemCS is assumed: $e_x = \{\frac{dx(\xi,\eta,\zeta)}{d\xi}, \frac{dy(\xi,\eta,\zeta)}{d\xi}, \frac{dz(\xi,\eta,\zeta)}{d\xi}\}, h = \{\frac{dx(\xi,\eta,\zeta)}{d\eta}, \frac{dy(\xi,\eta,\zeta)}{d\eta}, \frac{dz(\xi,\eta,\zeta)}{d\eta}\}, e_z = e_x \times h, e_y = e_z \times e_x$, where ξ, η, ζ are parametric element coordinates. The strains and stresses in individual layers are always reported in material coordinate system. LayeredCS integration The layered cross section integration can be set up using number of integrations points in layer plane (`nintegrationpoints` parameter) and using number of integration points per layer thickness (`layerintegrationpoints` parameter).

Keyword	**QWedge**
Description	3D wedge six-node finite element with quadratic interpolation
Specific parameters	[`NIP` $\#_{\rm (in)}$]
Parameters	`NIP`: allows to alter the default integration formula (possible completions are 2 (default) and 9 point integration formulas).
Unknowns	Three dofs (u-displacement, v-displacement, w-displacement) are required in each node.
Approximation	Quadratic approximation of displacements and geometry.
Integration	Full integration of all strain components using four point Gauss integration formula.
Features	Layered cross section support.
CS properties	-
Loads	-
Nlgeo	0,1,2.
Status	Reliable

Table 45: QWedge element summary

2.9 Interface elements

Interface elements represents an interaction between points, edges or surfaces. They are used to model debonding between surfaces or more general fracture processes through the use of cohesive zone (cz) models. They can also be used to model contact between elements. Specific interface material models needs to be used - consult the *matlibmanual* for supported models.

Ordering convention: All inerface elements have *plus*-side and a *minus*-side and all nodes should first be specified for the minus-side and then the plus-side. The normal to the interface is defined to point from the minus-side to the-plus side. Direction of normal vector on an element specifies normal stress/cohesion across the element. It is assumed that normal jump, normal traction are at the first position of corresponding vectors. Stiffness matrix in local coordinates has always on position 1,1 normal stiffness and then shear stiffness.

2.9.1 IntElPoint, Interface1d elements

Implementation of one dimensional (slip) interface element. This element connects two separate nodes and the interaction is governed by a one-dimensional slip law. This law determines the force acting between the nodes as a function on their relative displacement in the slip direction. The element can be used in 1D, 2D, and 3D (default) and its features are summarized in Table 46.

Keyword	**IntElPoint, Interface1d (deprecated)**
Description	One dimensional (slip) interface element
Specific parameters	[`refnode` $\#_{(in)}$] [`normal` $\#_{(ra)}$]
Parameters	`refnode`: determines the reference node, which is used to specify a reference direction (the direction vector is obtained by subtracting the coordinates of the first node from the reference node). `normal`: The reference direction can be directly specified by the optional parameter `normal`. Although both `refnode` and `normal` are optional, at least one of them must be specified.
Unknowns	One, two, or three DOFs (u-displacement, v-displacement, w-displacement) are required in each node, according to element mode (determined from domain type).
Approximation	-
Integration	-
Features	-
CS properties	-
Loads	-
Nlgeo	0
Status	Reliable
Note	Element requires material model with _1dInterface support.

Table 46: IntElPoint element summary

2.9.2 IntElLine1, Interface2dlin elements

Implementation of a two dimensional line element with a linear approximation of the displacement jump. The element can be used to tie together two element edges and is defined by four nodes - two on each edge. Note that, the nodes along the interface are doubled, each couple with identical coordinates. Nodes on the negative side are numbered first, followed by nodes on the positive part. Requires material model with _2dInterface support. The element features are summarized in Table 47.

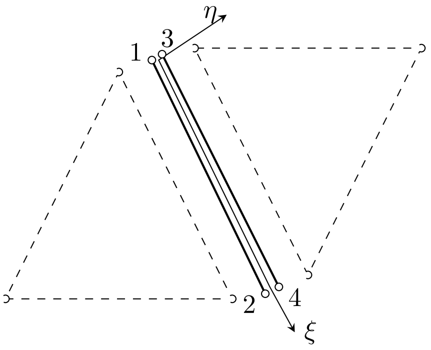

Figure 24: Interface2dlin element with linear interpolation. Definition and node numbering convention

Keyword	**IntElLine1, Interface2dlin(deprecated)**
Description	2D interface element with linear approximation
Specific parameters	[`axisymmode` #$_{(0)}$]
Parameters	`axisymmode`: Flag controlling axisymmetric mode (integration over unit circumferential angle).
Unknowns	Two dofs (u-displacement, v-displacement) are required in each node.
Approximation	Linear approximation of displacements and geometry.
Integration	Full integration of all strain components using two point integration formula.
Features	-
CS properties	-
Loads	-
Nlgeo	0
Status	Reliable
Note	Element requires material model with _2dInterface support.

Table 47: IntElLine1 element summary

2.9.3 IntElLine2, Interface2dquad elements

Implementation of a two dimensional interface element with quadratic approximation of displacement field. Can be used to glue together two elements with quadratic displacement approximation along the shared edge. Note, that the nodes along the interface are doubled, each couple with identical coordinates. Nodes on the negative side are numbered first, followed by nodes on the positive part. Requires material model with _2dInterface support. The element features are summarized in Table 48.

2.9.4 IntElSurfTr1, Interface3dtrlin elements

Implementation of a three dimensional interface element with linear approximation of displacement field. Can be used to glue together two elements with linear displacement approximation along the shared triangular surface. Note, that the nodes along the interface are doubled, each couple with identical coordinates. Nodes on the negative surface are numbered first, followed by nodes on the positive part. The numbering of surface nodes on positive surface (+) determines the positive normal (right hand rule). Requires material model with _3dInterface support. The element features are summarized in Table 49.

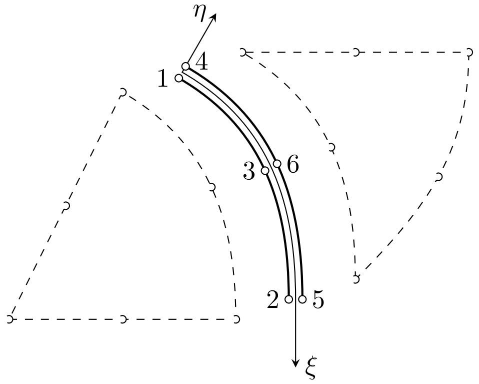

Figure 25: Interface2dquad element with quadratic interpolation. Definition and node numbering convention

Keyword	**IntElLine2, Interface2dquad (deprecated)**
Description	2D interface element with quadratic approximation
Specific parameters	[`axisymmode` #$_{(0)}$]
Parameters	`axisymmode`: Flag controlling axisymmetric mode (integration over unit circumferential angle).
Unknowns	Two dofs (u-displacement, v-displacement) are required in each node.
Approximation	Quadratic approximation of displacements and geometry.
Integration	Full integration of all strain components using four point integration formula.
Features	-
CS properties	-
Loads	-
Nlgeo	0
Status	Reliable
Note	Element requires material model with _2dInterface support.

Table 48: IntElLine2 element summary

Keyword	**IntElSurfTr1, Interface3dtrlin (deprecated)**
Description	3D interface element with linear approximation
Specific parameters	-
Unknowns	Three dofs (u-displacement, v-displacement, w-displacement) are required in each node.
Approximation	Linear approximation of displacements and geometry.
Integration	Full integration of all components using one point integration formula.
Features	-
CS properties	-
Loads	-
Nlgeo	0
Status	Reliable
Note	Element requires material model with _3dInterface support.

Table 49: IntElSurfTr1 element summary

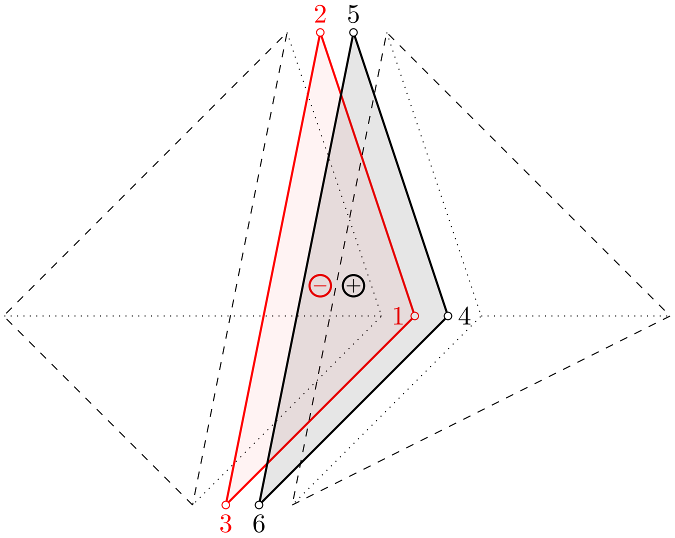

Figure 26: Interface3dtrlin element with linear interpolation. Definition and node numbering convention

2.9.5 IntElSurfQuad1 element

Implementation of a three dimensional interface element with linear approximation of displacement field. Can be used to glue together two elements with linear displacement approximation along the shared quad surface. Note, that the nodes along the interface are doubled, each couple with identical coordinates. Nodes on the negative surface are numbered first, followed by nodes on the positive part. The numbering of surface nodes on positive surface (+) determines the positive normal (right hand rule). Requires material model with _3dInterface support. The element features are summarized in Table 50.

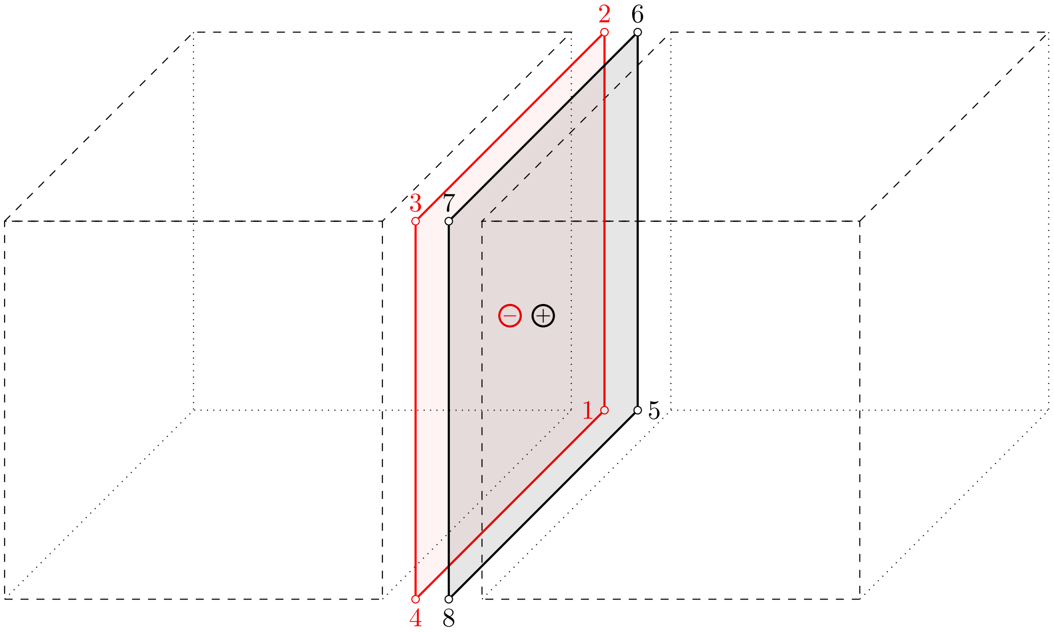

Figure 27: IntElSurfQuad1 element with linear interpolation. Definition and node numbering convention

Keyword	**IntElSurfQuad1**
Description	3D interface element with linear approximation
Specific parameters	-
Unknowns	Three dofs (u-displacement, v-displacement, w-displacement) are required in each node.
Approximation	Linear approximation of displacements and geometry.
Integration	Full integration of all components using one point integration formula.
Features	-
CS properties	-
Loads	-
Nlgeo	0
Status	Reliable
Note	Element requires material model with _3dInterface support.

Table 50: IntElSurfQuad1 element summary

2.9.6 Bondlink3d element

This class implements a bond link for connecting beam (frame) and continuum elements in unstructured meshes. The main idea is to use the rotation of the beam element and the rigid arm from the beam node to the continuum element node to compute the displacement jump along the rebar element (and two components, which are perpendicular to each other and lie in a plane for which the direction along the rebar is normal to. This element differs from the lattice link element, for which both the beam and the lattice nodes have rotational DOFs and therefore the lattice node's rotations can be used to compute the displacement jump at the beam element location. It also differs from the other interface elements since the first stiffness component is in the direction of the slip.

This element represents a two-node 3d link element connecting 3d beam and 3d lattice elements. The bond area for this element is calculated from the length and diameter of the truss element that is bonded to the continuum as $A_{\text{bond}} = \pi d l$, where A_{bond} is the bond area, d is the reinforcement diameter and l is the length of the element. Each node has six degrees of freedom. The input parameters for this element are shown in Table 2.9.6.

Keyword	**bondlink3d**
Description	3d bond link element for beams
Specific parameters	`length` #$_{\rm (rn)}$ `diameter` #$_{\rm (rn)}$ `dirvector` #$_{\rm (ra)}$ `l_end` #$_{\rm (rn)}$
Parameters	`length`: bond length `diameter`: diameter `dirvector`: direction vector in which bond-slip occurs.
Unknowns	Six dofs (u-displacement, v-displacement, w-displacement, u-rotation, v-rotation and w-rotation) are required in the beam node and three dofs (u-displacement, v-displacement, w-displacement) in the continnum node. The order of input of nodes must be beam node first and continuum node second.
Reference	[10]

Table 51: latticelink3d element summary

2.9.7 Bondlink3dtruss element

This class implements a bond link for connecting trusses and continuum elements for meshes in which the nodes of the trusses and continuum elements coincide. The bond area for this element is calculated from the length and diameter of the truss element that is bonded to the continuum as $A_{\text{bond}} = \pi d l$, where A_{bond} is the bond area, d is the reinforcement diameter and l is the length of the element. The input parameters for this element are shown in Table 52.

Keyword	**bondlink3dtruss**
Description	3d bond link element for trusses
Specific parameters	`length` $\#_{\text{(rn)}}$ `diameter` $\#_{\text{(rn)}}$ `dirvector` $\#_{\text{(ra)}}$
Parameters	`length`: bond length `diameter`: diameter of rebar modelled by truss element `dirvector`: direction vector in which bond-slip occurs.
Unknowns	Three dofs (u-displacement, v-displacement, w-displacement) are required in each node.

Table 52: bondlink3dtruss element summary

2.10 Free warping analysis elements

2.10.1 TrWarp

Implements 2D linear triangular three-node finite element for `FreeWarping` analysis. Each node has 1 degree of freedom. The node numbering is anti-clockwise. The element features are summarized in Table 53.

Keyword	**TrWarp**
Description	2D linear triangular warping element
Specific parameters	-
Unknowns	One dof (the value of deplanation function) is required in each node.
Approximation	Linear approximation of displacements and geometry.
Integration	Integration using one point gauss integration formula.
Features	This type of element is supported in `FreeWarping` analysis only.
CS properties	Only `warpingCS` is supported.
Loads	Edge loads corresponding to free warping problem are generated automatically. Additional edge loads are not supported.
Nlgeo	0.
Status	-
Note	Test case: sm/freewarpingtest2.in

Table 53: TrWarp element summary

2.11 XFEM elements

XFEM elements allow simulations where the unknown fields are enriched through the partition of unity concept. Two elements are currently available for 2D XFEM simulations: *TrPlaneStress2dXFEM* (subclass of *TrPlaneStress2d*) and *PlaneStress2dXfem* (subclass of PlaneStress2d).

2.11.1 TrPlaneStress2dXFEM element

Keyword	**TrPlaneStress2dXFEM**
Description	Two dimensional 3-node triangular XFEM element
Specific parameters	[`czmaterial` $\#_{(\mathrm{in})}$] [`nipcz` $\#_{(\mathrm{in})}$][`useplanestrain` $\#_{(\mathrm{in})}$]
Parameters	`czmaterial`: Interface material for the cohesive zone. (If no material is specified, traction free crack surfaces are assumed.) `nipcz`: Number of integration points used on each segment of the cohesive zone. `useplanestrain`: If plane strain or plane stress should be assumed. 0 implies plane stress and 1 implies plane strain.
Unknowns	Two continuous (standard) DOFs (u-displacement, v-displacement) and a variable number of enriched DOFs (can be continuous or discontinuous).
Approximation	-
Integration	Elements cut by an XFEM interface are divided into subtriangles.
Features	-
CS properties	-
Loads	-
Nlgeo	-
Status	-
Note	Test case: sm/xFemCrackVal.in

Table 54: TrPlaneStress2dXFEM element summary

2.11.2 PlaneStress2dXfem element

Keyword	**PlaneStress2dXfem**
Description	Two dimensional 4-node quad XFEM element
Specific parameters	[`czmaterial` #(in)] [`nipcz` #(in)][`useplanestrain` #(in)]
Parameters	`czmaterial`: Interface material for the cohesive zone. (If no material is specified, traction free crack surfaces are assumed.) `nipcz`: Number of integration points used on each segment of the cohesive zone. `useplanestrain`: If plane strain or plane stress should be assumed. 0 implies plane stress and 1 implies plane strain.
Unknowns	Two continuous (standard) DOFs (u-displacement, v-displacement) and a variable number of enriched DOFs (can be continuous or discontinuous).
Approximation	-
Integration	Elements cut by an XFEM interface are divided into subtriangles.
Features	-
CS properties	-
Loads	-
Nlgeo	-
Status	-
Note	Test cases: sm/xFemCrackValBranch.in, sm/xfemCohesiveZone1.in, benchmark/xfem01.in

Table 55: PlaneStress2dXfem element summary

2.12 Iso Geometric Analysis based (IGA) elements

The following record describes the common part of IGA element record:

```
*IGAElement   (num#)(in)                                                  \
              mat #(in) crossSect #(in) nodes #(ia)                       \
              knotvectoru #(ra) knotvectorv #(ra) knotvectorw #(ra)       \
              [knotmultiplicityu #(ia)] [knotmultiplicityv #(ia)]         \
              [knotmultiplicityw #(ia)]                                   \
              degree #(ia) nip #(ia)                                      \
              ⟨[partitions #(ia)]⟩ ⟨[remote #()]⟩
```

The `knotvectoru`, `knotvectorv`, and `knotvectorw` parameters specify knot vectors in individual parametric directions, considering only distinct knots. Open knot vector is always assumed, so the multiplicity of the first and last knot should be equal to $p+1$, where p is polynomial degree in coresponding direction (determined by `degree` parameter, see further).

The knot multiplicity can be set using optional parameters `knotmultiplicityu`, `knotmultiplicityv`, and `knotmultiplicityw`. By default, the open knot vector is assumed and multiplicity of internal knots is assumed to be equal to one. Note, that total number of knots in particular direction (including multiplicity) must be equal to number of control points in this direction increased by degree in this direction plus 1.

The degree of approximation for each parametric direction is determined from `degree` array, dimension of which is equal to number of spatial dimensions of the problem.

In case of elements with BSpline or Nurbs interpolation, the nodes forming the rectangular array of control points of the element are ordered in a such way, that u-index is changing most quickly, and w-index (or v-index in case of 2d problems) most slowly. In case of elements with T-spline interpolation, the nodes forming the T-mesh of the element are ordered arbitrarily.

The supported ***IGAElement** values are following:

Keyword: `bsplineplanestresselement`
Parameters: None.
Keyword: `nurbsplanestresselement`
Parameters: None.
Keyword: `nurbs3delement`
Parameters: None.
Keyword: `tsplineplanestresselement`
Parameters: `localindexknotvectoru` #(in) `localindexknotvectorv` #(in) `localindexknotvectorw` #(in)

The parameters `localindexknotvectoru`, `localindexknotvectorv`,
and `localindexknotvectorw` defined by the indices to global knot vectors (given by `knotvectoru`, `knotvectorv`, and `knotvectorw` parameters) specify the local knot vectors for each control point of T-mesh (node) in the same order as the nodes have been specified for the element. The local knot vector in a particular direction has $p+2$ entries, where the p is the polynomial degree in that direction.

2.13 Special elements

2.13.1 LumpedMass element

This element, defined by a single node, allows to introduce additional concentrated mass and/or rotational inertias in a node. A different mass and rotary inertia may be assigned to each coordinate direction. At present, individual mass/inertia components can be specified for every degree of freedom of element node. Only displacement and rotational degrees od freedom are considered. The element features are summarized in Table 56.

Keyword	**LumpedMass**
Description	Lumped mass element
Specific parameters	`components` $\#_{\rm (ra)}$
Parameters	`components`: allows to specify additional concentrated mass components (Force*Time^2/Length) and rotary inertias (Force*Length*Time^2) about the nodal coordinate axes. `dofs`: dofs to which the components apply.
Unknowns	As specified by `dofs`.
Approximation	-
Integration	-
Features	-
CS properties	-
Loads	-
Status	Reliable

Table 56: LumpedMass element summary

2.13.2 Spring element

This element represent longitudial or torsional spring element. It is defined by two nodes, orientation and a spring constant. The spring element has no mass associated, the mass can be added using LumpedMass element. The spring is linear and works the same way in tension or in compression. The element features are summarized in Table 57.

Keyword	**Spring**
Description	Spring element
Specific parameters	`mode` $\#_{\rm (in)}$ `k` $\#_{\rm (rn)}$ [`m` $\#_{\rm (rn)}$] `orientation` $\#_{\rm (ra)}$
Parameters	`mode`: defines the type of spring element (see Table 58). `k`: determines the spring constant, corresponding units are [Force/Length] for longitudinal spring and [Force*Length/Radian] for torsional spring. `orientation`:defines orientation vector of spring element (of size 3) - for longitudinal spring it defines the direction of spring, for torsional spring it defines the axis of rotation. `m`: determines optional mass of the element, zero value assumed by default.
Note	the spring element nodes doesn't need to be coincident, but the spring orientation is always determined by `orientation` vector.

Table 57: Spring element summary

mode	description
0	1D spring element along x-axis, requires D_u DOF in each node, orientation vector is {1,0,0}
1	2D spring element in xy plane, requires D_u and D_v DOFs in each node (orientation vector should be in xy plane)
2	2D spring element in xz plane, requires D_u and D_w DOFs in each node (orientation vector should be in xz plane)
3	2D torsional spring element in xz plane, requires R_v DOFs in each node
4	3D spring element in space, requires D_u, D_v, and D_w DOFs in each node
5	3D torsional spring in space, requires R_u, R_v, and R_w DOFs in each node

Table 58: Supported spring element modes

nlgeo	strain tensor
0 (default)	Small-strain tensor
1	Green-Lagrange strain tensor
2	Deformation gradient

Table 59: Nonlinear geometry modes

2.14 Geometric nonlinear analysis

To take int account geometric nonlinearity for a specific element, the keyword `nlgeo` must be specified. The `nlgeo` parameter defines which formulation of the momentum balance is solved and what deformation measure that is computed and sent to the consitiutive models (see Table 59). If `nlgeo`=1, then the momentum balance is set up in the reference configuration in terms of the First Piola-Kirchhoff stress tensor $\mathbf{P}$ and the deformation tensor $\mathbf{F}$ as energy conjugates. This is also refered to a Total Lagrangian formulation. The balance equation in weak form reads

$$\int_\Omega \delta\mathbf{F} : \mathbf{P} \mathrm{d}\Omega = \int_\Gamma \delta\mathbf{x} \cdot \mathbf{t}_P \mathrm{d}\Gamma + \int_{\partial\Omega} \delta\mathbf{x} \cdot \mathbf{b}_P \mathrm{d}\Omega \tag{7}$$

This equation can be rewritten in terms of the displacement $\mathbf{u}$ and the displacement gradient $\mathbf{H}$

$$\int_\Omega \delta\mathbf{H} : \mathbf{P} \mathrm{d}\Omega = \int_\Gamma \delta\mathbf{u} \cdot \mathbf{t}_P \mathrm{d}\Gamma + \int_{\partial\Omega} \delta\mathbf{u} \cdot \mathbf{b}_P \mathrm{d}\Omega \tag{8}$$

This equation is nearly identical to the one for small strains except that another stress measure is used and we have the virtual displacement gradient instead of the virtual strains.

The corresponding FE-formulation is obtained as

$$\int_\Omega \mathbf{B}_H^{\mathrm{T}} \cdot \mathbf{P} \mathrm{d}\Omega = \int_\Gamma \mathbf{N}^{\mathrm{T}} \cdot \mathbf{t}_P \mathrm{d}\Gamma + \int_{\partial\Omega} \delta\mathbf{N}^{\mathrm{T}} \cdot \mathbf{b}_P \mathrm{d}\Omega \tag{9}$$

with the tangent stiffness

$$\mathbf{K}_{\mathrm{T}} = \int_\Omega \mathbf{B}_H^{\mathrm{T}} \cdot \frac{\partial\mathbf{P}}{\partial\mathbf{F}} \cdot \mathbf{B}_H \mathrm{d}\Omega \tag{10}$$

Thus, for an element to support large deformations (in addition to small deformation) it needs only to implement the $\mathbf{B}_H$ matrix. Similar to the regular $\mathbf{B}$ matrix, which gives the strains in Voigt form when multiplied with the solution vector $\mathbf{a}$, $\mathbf{B}_H$ should give the displacement gradient in Voigt form with 9 components for a full 3D state.

3 Elements for Transport problems (TM Module)

3.1 1D Elements

3.1.1 Line1ht element

Two node linear isoparametric element for for heat transfer problems. Each node has 1 degree of freedom. The cross section property "area" is requested from the cross section. The element geometry is specified in (x,y,z) plane. The element features are summarized in Table 60. Stabilization through lumped capacity matrix is suggested in transient transport problems, using **lumped**.

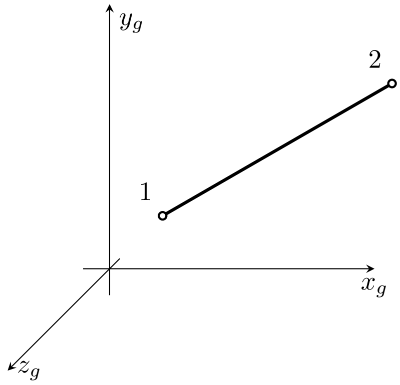

Figure 28: Line1ht element in (x,y,z) space.

Keyword	**Line1ht**
Description	Line (truss-like) finite element with linear approximation for heat transfer problems
Specific parameters	-
Unknowns	Single dof (T_f - temperature) is required in each node.
Approximation	Linear approximation of temperature.
Integration	Integration using one point gauss integration formula.
Loads	Body loads are supported. Boundary loads are unsupported.
Features	-
CS properties	Area
Status	Reliable

Table 60: Line1ht element summary

3.1.2 Line1mt element

Line (truss-like) finite element with linear approximation of moisture. Other features are the same as for Line1ht in Section 3.1.1.

3.1.3 Line1hmt element

Line (truss-like) finite element with linear approximations of temperature and moisture. Other features are the same as for Tr1ht in Section 3.1.1.

3.2 2D Elements

3.2.1 Tr1ht element

Implements the linear triangular finite element for heat transfer problems. Each node has 1 degree of freedom. The cross section thickness property is requested form cross section model. The node numbering is anti-clockwise. The element features are summarized in Table 61.

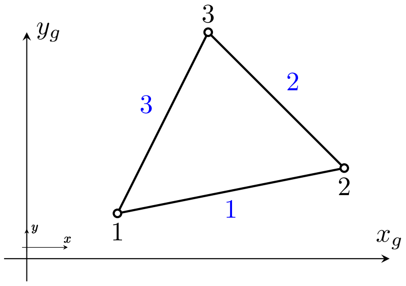

Figure 29: Tr1ht element - node and side numbering.

Keyword	**Tr1ht**
Description	triangular finite element with linear approximation for heat transfer problems
Specific parameters	-
Unknowns	Single dof (T_f - temperature) is required in each node.
Approximation	Linear approximation of temperature.
Integration	Integration using one point gauss integration formula.
Loads	Body loads are supported. Boundary loads are supported and are computed using numerical integration. The side numbering is following. Each i-th element side begins in i-th element node and ends on next element node (i+1-th node or 1-st node, in the case of side number 3). The local positive edge x-axis coincides with side direction, the positive local edge y-axis is rotated 90 degrees anti-clockwise (see fig. (29)).
Features	-
CS properties	-
Status	

Table 61: Tr1ht element summary

3.2.2 Tr1mt element

Isoparametric triangular finite element with linear approximation of moisture. Other features are the same as for Tr1ht in Section 3.2.1.

3.2.3 Tr1hmt element

Isoparametric triangular finite element with linear approximations of temperature and moisture. Other features are the same as for Tr1ht in Section 3.2.1.

3.2.4 Quad1ht element

Represents isoparametric four-node quadrilateral finite element for heat transfer problems. Each node has 1 degree of freedom. Problem should be defined in x,y plane. The cross section thickness property is requested form cross section model. The nodes should be numbered anti-clockwise (positive rotation around z-axis). The element features are summarized in Table 62.

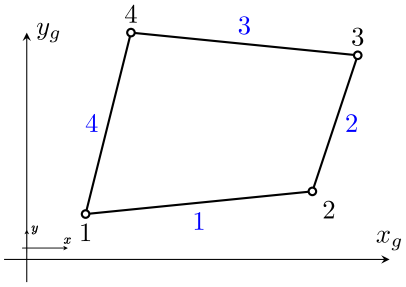

Figure 30: Quad1ht element. Node numbering, Side numbering and definition of local edge c.s.(a).

Keyword	**Quad1ht**
Description	Isoparametric four-node quadrilateral linear interpolation element for heat transfer problems
Specific parameters	[`NIP` #$_{(in)}$]
Parameters	`NIP`: allows to change the default number of integration point used.
Unknowns	Single dof (T_f - temperature) is required in each node.
Approximation	Linear approximation of temperature.
Integration	Integration using gauss integration formula in 4 (the default), 9, or 16 integration points. The default number of integration point used can be overloaded using `NIP` parameter.
Loads	Body loads are supported. Boundary loads are supported and computed using numerical integration. The side numbering is following. Each i-th element side begins in i-th element node and ends on next element node (i+1-th node or 1-st node, in the case of side number 4). The local positive edge x-axis coincides with side direction, the positive local edge y-axis is rotated 90 degrees anti-clockwise (see fig. (30)).
Features	-
CS properties	-
Status	

Table 62: Quad1ht element summary

3.2.5 Quad1mt element

Isoparametric four-node quadrilateral finite element. Other features are the same as for Quad1ht in Section 3.2.4.

3.2.6 Quad1hmt element

Represents isoparametric four-node quadrilateral finite element for heat and mass (one constituent) transport problems. Two dofs (T_f - temperature and C_1 - concentration) are required in each node. Linear approximation of temperature and mass concentration. Other features are similar to Quad1 element, see section 3.2.4.

3.2.7 QQuad1ht element

Represents isoparametric quadratic eight-node quadrilateral finite element for heat transfer problems. Each node has 1 degree of freedom. Problem should be defined in x,y plane. The cross section thickness property is requested form the cross section model. The nodes should be numbered anti-clockwise (positive rotation around z-axis), see fig. 12. The element has the same features as in Table 62.

3.2.8 QQuad1mt element

Element for mass transport problems, see the parent element in sec. 3.2.7.

3.2.9 QQuad1hmt element

Element for heat and mass transport problems, see the parent element in sec. 3.2.7.

3.3 Axisymmetric Elements

3.3.1 Quadaxisym1ht element

Isoparametric four-node quadrilateral finite element for axisymmetric heat transfer problems. The element description is similar to Quad1 element, see section 3.2.4.

3.3.2 Traxisym1ht element

Linear triangular finite element for axisymmetric heat transfer problems. The element description is similar to Tr1ht element, see section 3.2.1.

3.4 3D Elements

3.4.1 Tetrah1ht - tetrahedral 3D element

Represents isoparametric four-node tetrahedral element. Each node has 1 degree of freedom. The same numbering convection is adopted as in mechanics, see Fig. 20. The element features are summarized in Table 63.

Keyword	**Tetrah1ht**
Description	Isoparametric, four-node tetrahedral element with linear approximation for heat transfer problems
Specific parameters	[`NIP` #$_{\text{(in)}}$]
Parameters	`NIP`: allows to change the default number of integration point used.
Unknowns	Single dof (T_f - temperature) is required in each node.
Approximation	Linear approximation of temperature.
Integration	Integration using gauss integration formula in 1 (the default), or 4 integration points. The default number of integration point used can be overloaded using `NIP` parameter.
Loads	Body loads are supported. Boundary loads are supported and computed using numerical integration. The side and surface numbering is shown in Fig. 20.
Features	-
CS properties	-
Status	

Table 63: Tetrah1ht element summary

3.4.2 Brick1ht - hexahedral 3D element

Represents isoparametric eight-node brick/hexahedron finite element for heat transfer problems. Each node has 1 degree of freedom. The element features are summarized in Table 64.

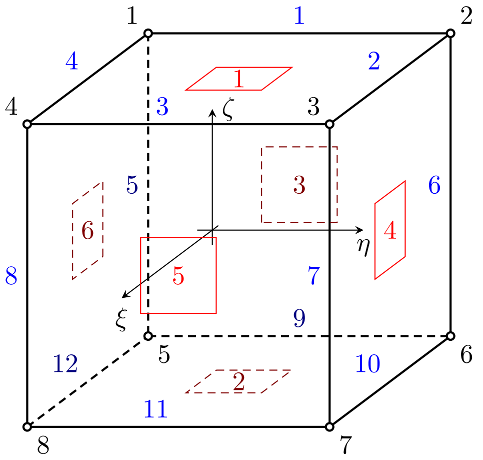

Figure 31: Brick1ht element. Node numbers are in black, side numbers are in blue, and surface numbers are in red.

Keyword	**Brick1ht**
Description	Isoparametric, hexahedral 3D element with linear approximation for heat transfer problems
Specific parameters	[`NIP` $\#_{\rm (in)}$]
Parameters	`NIP`: allows to change the default number of integration point used.
Unknowns	Single dof (T_f - temperature) is required in each node.
Approximation	Linear approximation of temperature.
Integration	Integration using gauss integration formula in 8 (the default), or 27 integration points. The default number of integration point used can be overloaded using `NIP` parameter.
Loads	Body loads are supported. Boundary loads are supported and computed using numerical integration. The side and surface numbering is shown in fig. (31)).
Features	-
CS properties	-
Status	

Table 64: Brick1ht element summary

3.4.3 Brick1hmt - hexahedral 3D element

Represents isoparametric eight-node quadrilateral finite element for heat and mass (one constituent) transfer problems. Two dofs (T_f - temperature and C_1 - concentration) are required in each node. Linear approximation of temperature and mass concentration. Other features are similar to Brick1 element, see section 3.4.2.

3.4.4 QBrick1ht - quadratic hexahedral 3D element

Implementation of quadratic 3d 20-node finite element. Each node has 1 degree of freedom. See section 2.8.3 for node numbering order and order of faces. The element features are summarized in Table 65.

Keyword	**QBrick1ht**
Description	Isoparametric, hexahedral 3D element with quadratic approximation for heat transfer problems
Specific parameters	[`NIP` $\#_{\rm (in)}$]
Parameters	`NIP`: allows to change the default number of integration point used, possible values are 8, 27 (default) and 64.
Unknowns	Single dof (T_f - temperature) is required in each node.
Approximation	Quadratic approximation of temperature and geometry..
Integration	Integration using gauss integration formula in 8, 27 (default), or 64 integration points. The default number of integration point used can be overloaded using `NIP` parameter.
Loads	-
Features	-
CS properties	-
Status	

Table 65: QBrick1ht element summary

3.4.5 QBrick1mt - quadratic hexahedral 3D element

The same element as QBrick1ht for mass transfer problems, see 3.4.4. Linear approximation of mass concentration.

3.4.6 QBrick1hmt - quadratic hexahedral 3D element

The same element as QBrick1ht for heat and mass (one constituent) transfer problems. Two dofs (T_f - temperature and C_1 - concentration) are required in each node. Linear approximation of temperature and mass concentration. Other features are similar to QBrick1ht element, see section 3.4.4.

3.5 Lattice transport elements

These are two-node lattice transport elements intended for coupled and stand-alone mass- or heat-transport analyses on Delaunay/Voronoi lattice networks. Each node carries one pressure (or temperature) degree of freedom. Flow along the element follows a linear pressure interpolation; the cross-section polygon is the dual Voronoi facet and enters strength/stiffness-like quantities through `polycoords`.

The element assembles a capacity matrix that can optionally be switched from the default consistent form to a diagonal (row-sum-preserving) lumped form. Lumping makes the scheme equivalent to a two-point-flux-approximation (TPFA) finite-volume discretisation on the Voronoi dual and is recommended when the capacity $c(p)$ is strongly nonlinear (e.g. drying, Richards-type problems), where the consistent form is known to produce pressure oscillations and an apparent loss of conductivity near sharp fronts. Lumping is opt-in via `lumpedcapacity`; the default preserves existing behaviour.

3.5.1 latticemt2d element

Two-node 2D lattice mass-transport element. The element features are summarised in Table 66.

Keyword	**latticemt2d**
Description	2d lattice mass-transport element
Specific parameters	`thick` #(rn) `width` #(rn) `gpcoords` #(ra) [`dim` #(rn)] [`crackwidth` #(rn)] [`couplingflag` #(in)] [`couplingnumber` #(in)] [`lumpedcapacity` #(in)]
Parameters	`thick`: out-of-plane thickness. `width`: Voronoi-facet width (cross-section length). `gpcoords`: integration-point coordinates in the global system. `dim`: dimension factor (optional, default 2). `crackwidth`: crack width used in conductivity scaling (optional). `couplingflag`: flag (optional, default 0) activating coupling with a mechanical lattice element. `couplingnumber`: number of the coupled mechanical element. `lumpedcapacity`: flag (optional, default 0). If set to 1, the capacity matrix is replaced by its diagonal (row-sum-preserving) lumped form, yielding a TPFA-equivalent, monotone scheme. Recommended for strongly nonlinear $c(p)$.
Unknowns	Single dof (P_f - pressure / moisture) per node.
Approximation	Linear pressure along the element.
Integration	One Gauss point at `gpcoords`.
CS properties	simplecs.
Status	Reliable

Table 66: latticemt2d element summary

3.5.2 latticemt3d element

Two-node 3D lattice mass-transport element. The cross-section is a polygonal Voronoi facet supplied via `polycoords`. The element features are summarised in Table 67.

Keyword	**latticemt3d**
Description	3d lattice mass-transport element
Specific parameters	`polycoords` #(ra) [`dim` #(rn)] [`area` #(rn)] [`mlength` #(rn)] [`crackwidths` #(ra)] [`couplingflag` #(in)] [`couplingnumber` #(ia)] [`lumpedcapacity` #(in)]
Parameters	`polycoords`: coordinates of the mid-cross-section (Voronoi facet) vertices in the global system. `dim`: dimension factor (optional, default 3). `area`: prescribed cross-section area (optional; otherwise computed from `polycoords`). `mlength`: minimum element length threshold (optional). `crackwidths`: per-vertex crack widths used in conductivity scaling (optional). `couplingflag`: flag (optional, default 0) activating coupling with a mechanical lattice element. `couplingnumber`: array of coupled mechanical element numbers. `lumpedcapacity`: flag (optional, default 0). If set to 1, the capacity matrix is replaced by its diagonal (row-sum-preserving) lumped form, yielding a TPFA-equivalent, monotone scheme. Recommended for strongly nonlinear $c(p)$.
Unknowns	Single dof (P_f - pressure / moisture) per node.
Approximation	Linear pressure along the element.
Integration	One Gauss point at the element midpoint.
CS properties	simplecs.
Status	Reliable

Table 67: latticemt3d element summary

3.5.3 latticemt3dboundary element

Three-noded 3D lattice mass-transport element for boundaries of 3D periodic cells. The first two nodes carry the single pressure / moisture dof as in latticemt3d; the third is a control node that supplies the macro gradients driving the periodic cell. The dof of the node that lies outside the periodic cell is reconstructed from its periodic image inside the cell plus the control-node dofs, using the same translation pattern as lattice3dboundary (Figure 8). The element features are summarised in Table 68.

Keyword	**latticemt3dboundary**
Description	3d lattice mass-transport boundary element
Specific parameters	`polycoords` #$_{(ra)}$ `location` #$_{(ia)}$ [`dim` #$_{(rn)}$] [`area` #$_{(rn)}$] [`mlength` #$_{(rn)}$] [`crackwidths` #$_{(ra)}$] [`couplingflag` #$_{(in)}$] [`couplingnumber` #$_{(ia)}$] [`lumpedcapacity` #$_{(in)}$]
Parameters	`polycoords`: coordinates of the mid-cross-section (Voronoi facet) vertices in the global system. `location`: array of two integers between 1 and 26 specifying the location of the two transport nodes with respect to the 3D periodic cell — same encoding as lattice3dboundary. `dim`: dimension factor (optional, default 3). `area`: prescribed cross-section area (optional; otherwise computed from `polycoords`). `mlength`: minimum element length threshold (optional). `crackwidths`: per-vertex crack widths used in conductivity scaling (optional). `couplingflag`: flag (optional, default 0) activating coupling with a mechanical lattice element. `couplingnumber`: array of coupled mechanical element numbers. `lumpedcapacity`: flag (optional, default 0). Replaces the capacity matrix with its diagonal lumped form for monotone TPFA-equivalent behaviour.
Unknowns	Single dof (P_f) per transport node. The third (control) node carries the periodic-cell dofs as for lattice3dboundary but reduced to the transport-relevant components.
Approximation	Linear pressure along the element.
Integration	One Gauss point at the element midpoint.
CS properties	simplecs.
Status	Reliable

Table 68: latticemt3dboundary element summary

4 Elements for Fluid Dynamics problems (FM Module)

4.1 Stokes' Flow Elements

Stokes' flow elements neglect acceleration, and thus requires no additional stabilization.

4.1.1 Tr21Stokes element

Standard 6 node triangular element for stokes flow, with quadratic geometry, velocity and linear pressure. Both compressible and incompressible material behavior is supported (and also the seamless transition between the two). The element features are summarized in Table 69.

Keyword	**Tr121Stokes**
Description	Standard 6 node triangular element for stokes flow, with quadratic geometry, velocity and linear pressure
Specific parameters	-
Unknowns	Unknown pressure in nodes 1–3 with unknown velocity (V_u and V_v) in all 6 nodes.
Approximation	Quadratic approximation of geometry and velocity, linear pressure approximation.
Integration	
Features	-
Status	Reliable

Table 69: Tr121Stokes element summary

4.1.2 Tet21Stokes element

Standard 10 node tetrahedral element for stokes flow, with quadratic geometry, velocity and linear pressure. The element features are summarized in Table 70.

Keyword	**Tet21Stokes**
Description	Standard 10 node tetrahedral element for stokes flow, with quadratic geometry, velocity and linear pressure
Specific parameters	-
Unknowns	Unknown pressure (P_f) in nodes 1–4, and unknown velocity (V_u, V_v, V_w) in all nodes.
Approximation	Quadratic approximation of geometry and velocity, linear pressure approximation.
Integration	
Features	-
Status	Untested

Table 70: Tet21Stokes element summary

4.1.3 Hexa21Stokes element

Standard 27 node hexahedral element for stokes flow, with quadratic geometry, velocity and linear pressure. The element features are summarized in Table 71.

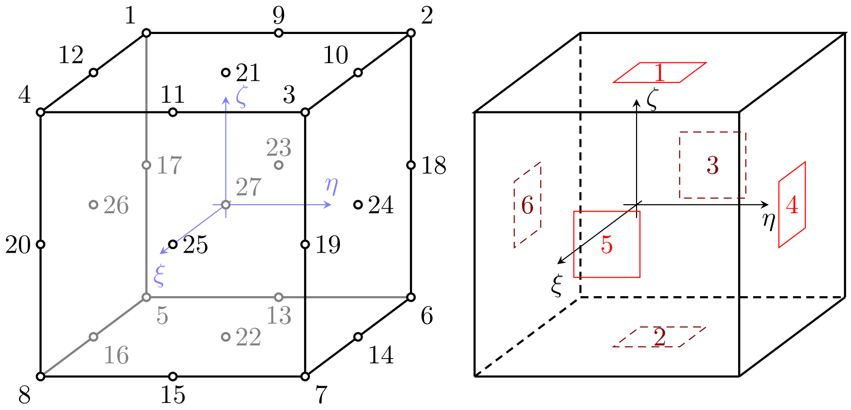

Figure 32: Hexa21Stokes element. Node numbering and face numbering.

Keyword	**Hexa21Stokes**
Description	Standard 10 node tetrahedral element for stokes flow, with quadratic geometry, velocity and linear pressure
Specific parameters	-
Unknowns	Unknown pressure (P_f) in nodes 1–8, and unknown velocity (V_u, V_v, V_w) in all nodes.
Approximation	Quadratic approximation of geometry and velocity, linear pressure approximation.
Integration	
Features	-
Status	Untested

Table 71: Hexa21Stokes element summary

4.1.4 Tr1BubbleStokes element

So called “Mini” element in 2D. A 3 node triangular element for stokes flow, with linear geometry and pressure. Velocity is enriched by a bubble function. Should not be used with materials that have memory (which is uncommon for flow problems). The element features are summarized in Table 72.

Keyword	**Tr1BubbleStokes**
Description	So called “Mini” 2D element
Specific parameters	-
Unknowns	Unknown pressure (P_f) in all nodes, and unknown velocity (V_u, V_v) in all nodes and one internal dof manager.
Approximation	Linear geometry and pressure. Velocity is enriched by a bubble function.
Integration	
Features	-
Status	Untested

Table 72: Tr1BubbleStokes element summary

4.1.5 Tet1BubbleStokes element

So called “Mini” element in 3D. A 4 node tetrahedral element for stokes flow, with linear geometry and pressure. Velocity is enriched by a bubble function. Should not be used with materials that have memory (which is uncommon for flow problems). The element features are summarized in Table 73.

Keyword	**Tet1BubbleStokes**
Description	So called “Mini” 3D element
Specific parameters	-
Unknowns	Unknown pressure (P_f) in all nodes, and unknown velocity (V_u, V_v) in all nodes and one internal dof manager.
Approximation	Linear geometry and pressure. Velocity is enriched by a bubble function.
Integration	
Features	-
Status	Untested

Table 73: Tet1BubbleStokes element summary

4.2 2D CBS Elements

4.2.1 Tr1CBS element

Represents the linear triangular finite element for transient incompressible flow analysis using cbs algorithm with equal order approximation of velocity and pressure fields. Each node has 3 degrees of freedoms (two components of velocity and pressure). The node numbering is anti-clockwise. The element features are summarized in Table 74.

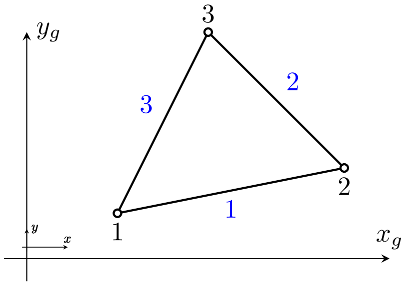

Figure 33: Tr1CBS element. Node numbering, Side numbering and definition of local edge c.s.(a).

Keyword	**Tr1CBS**
Description	linear triangular finite element for transient incompressible flow analysis using cbs algorithm
Specific parameters	[`bsides` #$_{(ia)}$] [`bcodes` #$_{(ia)}$]
Parameters	Since the problem formulation requires to evaluate some boundary terms, the element boundary edges should be specified as well as the types of boundary conditions applied at these boundary edges. The boundary edges (their numbers) are specified using `bsides` array. The type of boundary condition(s) applied to corresponding boundary side is determined by `bcodes` array. The available/supported boundary codes are following: 1 for prescribed traction, 2 for prescribed normal velocity, 4 for prescribed tangential velocity, and 8 for prescribed pressure. If the element side is subjected to a combination of these fundamental types boundary conditions, the corresponding code is obtained by summing up the corresponding codes.
Unknowns	Two velocity components (V_u and V_v) and pressure (P_f) are required in each node.
Approximation	Equal order approximation of velocity and pressure fields.
Integration	exact
Features	Constant boundary tractions are supported[1]. Body loads representing the self-weight load are supported.
Status	Untested

Table 74: Tr1CBS element summary

4.3 2D SUPG/PSGP Elements

4.3.1 Tr1SUPG element

Represents the linear triangular finite element for transient incompressible flow analysis using SUPG/PSPG stabilization with equal order approximation of velocity and pressure fields. Each node has 3 degrees of freedoms (two components of velocity and pressure). The node numbering is anti-clockwise. The element features are summarized in Table 75.

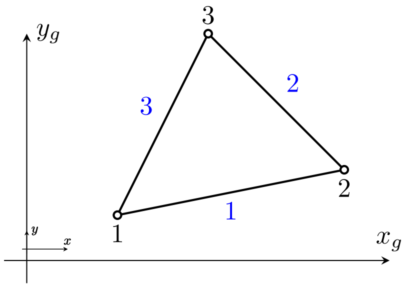

Figure 34: Tr1SUPG element. Node numbering, Side numbering and definition of local edge c.s.(a).

Keyword	**Tr1SUPG**
Description	linear triangular finite element for transient incompressible flow analysis using SUPG/PSPG algorithm
Specific parameters	[`vof` #(rn)] [`pvof` #(rn)]
Unknowns	Two velocity components (V_u and V_v) and pressure (P_f) are required in each node.
Approximation	Linear approximation of velocity and pressure fields.
Integration	exact
Loads	Constant boundary tractions are supported. Body loads representing the self-weight load are supported.
Multi-fluid analysis	The element has support for solving problems with two immiscible fluids in a fixed spatial domain. In the present implementation, a VOF and LevelSet tracking algorithms are used to track the position of interface. In case of VOF tracking, an initial VOF fraction (volume fraction of reference fluid) can be specified using `vof` (default is zero). Element can also be marked as allways filled with reference fluid (some form of source) using parameter `pvof` which specifies the permanent VOF value. In case of LevelSet tracking, the initial levelset is specified using reference polygon (see corresponding levelset record in oofem input manual). The material model should be of type **Keyword**: `twofluidmat`, that supports modelling of two immiscible fluids.
Status	Reliable

Table 75: Tr1SUPG element summary

4.3.2 Tr21SUPG element

Implementation of P2P1 Taylor Hood element for transient incompressible flow analysis using SUPG and LSIC stabilization. It consists of globally continuous, piecewise quadratic functions for approximation in velocity space and globally continuous, piecewise linear functions for approximation in pressure space. LBB condition is satisfied. There are 3 degrees of freedom in vertices (two components of velocity and pressure), and 2 degrees of freedom in edge nodes (two components of velocity only). The node numbering is anti-clockwise, vertices are numbered first. The element features are summarized in Table 76.

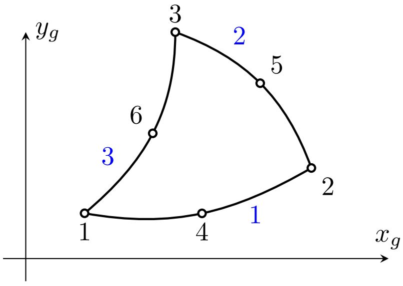

Figure 35: Tr21SUPG element - node and side numbering.

Keyword	**Tr21SUPG**
Description	P2P1 Taylor Hood element
Specific parameters	-
Unknowns	Two velocity components (V_u and V_v) and pressure (P_f) in vertices and two velocity components (V_u and V_v) in edge nodes are required.
Approximation	Quadratic approximation of velocity and linear approximation of pressure fields.
Integration	Integration is exact, each submatrix of element stiffness matrix is evaluated in proper number of Gauss points. Submatrices connected with velocity are evaluated in 7 or 13 points, mixed velocity-pressure submatrices in 3 or 7 points, submatrices connected with pressure in 3 points.
Loads	Constant boundary tractions are supported. Body loads representing the self-weight load are supported.
Multi-fluid analysis	The element has no support for solving problems with two immiscible fluids in a fixed spatial domain.
Status	Reliable

Table 76: Tr21SUPG element summary

4.3.3 Tr1SUPGAxi element

Represents the linear triangular finite element for transient incompressible flow analysis using SUPG/PSPG stabilization with equal order approximation of velocity and pressure fields in 2d-axisymmetric setting. Each node has 3 degrees of freedoms (two components of velocity and pressure). The y-axis is axis of ratational symmetry. The node numbering is anti-clockwise. The element features are summarized in Table 77.

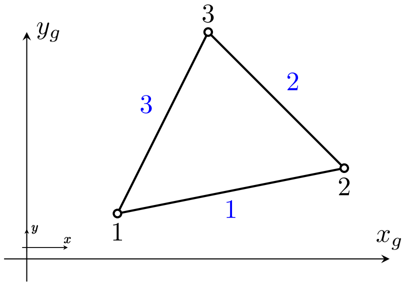

Figure 36: Tr1SUPGAxi element. Node numbering, Side numbering and definition of local edge c.s.(a).

Keyword	**Tr1SUPGAxi**
Description	linear equal order approximation axisymmetric element
Specific parameters	[`vof` #$_{\text{(rn)}}$][`pvof` #$_{\text{(rn)}}$]
Unknowns	Two velocity components (V_u and V_v) and pressure (P_f) are required in each node.
Approximation	Linear approximation of velocity and pressure fields.
Integration	Gauss integration in seven point employed.
Loads	Constant boundary tractions are supported. Body loads representing the self-weight load are supported.
Multi-fluid analysis	The element has support for solving problems with two immiscible fluids in a fixed spatial domain. In the present implementation, a VOF tracking algorithm is used to track the position of interface. An initial VOF fraction (volume fraction of reference fluid) can be specified using `vof` (default is zero). Element can also be marked as always filled with reference fluid (some form of source) using parameter `pvof` which specifies the permanent VOF value. In this case, the material model should be of type **Keyword**: `twofluidmat`, that supports modelling of two immiscible fluids.
Status	

Table 77: Tr1SUPGAxi element summary

4.4 3D SUPG/PSGP Elements

4.4.1 Tet1_3D_SUPG element

Represents 3D linear pyramid element for transient incompressible flow analysis using SUPG/PSPG stabilization with equal order approximation of velocity and pressure fields. Each node has 3 degrees of freedoms (two components of velocity and pressure). The element features are summarized in Table 78.

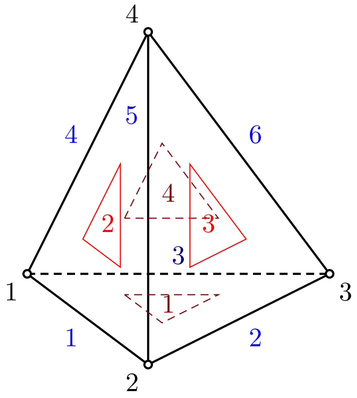

Figure 37: Tet1_3D_SUPG element.

Keyword	**TET1SUPG**
Description	linear equal order approximation axisymmetric element
Specific parameters	[`vof` #$_{\text{(rn)}}$][`pvof` #$_{\text{(rn)}}$]
Unknowns	Three velocity components (V_u, V_v, and V_w) and pressure (P_f) are required in each node.
Approximation	Linear approximation of velocity and pressure fields.
Integration	exact
Loads	Constant boundary tractions are supported. Body loads representing the self-weight load are supported.
Multi-fluid analysis	The element has support for solving problems with two immiscible fluids in a fixed spatial domain. In the present implementation, a LevelSet tracking algorithm is used to track the position of interface. The material model should be of type **Keyword**: `twofluidmat`, that supports modelling of two immiscible fluids.
Status	

Table 78: TET1SUPG element summary

References

[1] R. D. Cook and D. S. Malkus and M. E. Plesha, “Concepts and Applications of Finite Element Analysis”, Third Edition, isbn: 0-471-84788-7, 1989.

[2] Z. Bittnar and J. Sejnoha, “Numerical Methods in Structural Mechanics”,Thomas Telford,isbn:978-0784401705, 1996.

[3] R. Larsson and J. Mediavilla and M. Fagerström, “Dynamic fracture modeling in shell structures based on XFEM”, International Journal for Numerical Methods in Engineering, vol. 86, no. 4-5, 499–527, 2011.

[4] P. Grassl and M. Jirásek, "Meso-scale approach to modelling the fracture process zone of concrete subjected to uniaxial tension", International Journal of Solids and Structures, vol. 47, iss. 7-8, pp. 957-968, 2010..

[5] P. Grassl, J. Bolander, "Three-Dimensional Network Model for Coupling of Fracture and Mass Transport in Quasi-Brittle Geomaterials", Materials, 9, 782, 2016

[6] I. Athanasiadis, S. Wheeler and P. Grassl. "Hydro-mechanical network modelling of particulate composites", International Journal of Solids and Structures, vol. 130-131, pp. 49-60, 2018.

[7] P. Grassl and A. Antonelli. "3D network modelling of fracture processes in fibre-reinforced geomaterials", International Journal of Solids and Structures, vol. 156-157, pp. 234-242, 2019.

[8] Y. Toi. "Shifted integration technique in one-dimensional plastic collapse analysis using linear and cubic finite elements", International Journal for Numerical Methods in Engineering 31, no. 8, pp. 1537-1552, 1991.

[9] G. Abdelrhim and P. Grassl. "A simple frame element for large rotations", Available at SSRN 4850146, 2024.

[10] A. Sciegaj and P. Grassl and F. Larsson and K. Runesson and K. Lundgren. "Upscaling of three-dimensional reinforced concrete representative volume elements to effective beam and plate models", International Journal of Solids and Structures, vol. 202, pp. 835-853, 2020.